

# **CFD ANALYSIS OF NEWTONIAN FLUID FLOW PHENOMENA OVER A ROTATING CYLINDER**

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE  
REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology  
In  
Chemical Engineering**

Under the Guidance of  
**Prof. Basudeb Munshi**

**By Rohit Prabhakar  
(108CH043)**



**Department of Chemical Engineering  
National Institute of Technology  
Rourkela  
2012**

## National Institute of Technology, Rourkela



### CERTIFICATE

This is to certify that the thesis entitled, **“CFD ANALYSIS OF NEWTONIAN FLUID FLOW PHENOMENA OVER A ROTATING CYLINDER”** submitted by **Rohit Prabhakar** in partial fulfilments for the requirements for the award of Bachelor of Technology Degree in Chemical Engineering at National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by them under my supervision and guidance.

To the best of my knowledge, the matter embodied in this thesis has not been submitted to any other University / Institute for the award of any Degree or Diploma.

Date: 07-05-2012

---

Prof. Basudeb Munshi  
Dept. of Chemical Engineering  
National Institute of Technology  
Rourkela – 769008

## **ACKNOWLEDGEMENT**

I express my deepest appreciation and sincere gratitude to Prof. Basudeb Munshi for his valuable guidance, constructive criticism and timely suggestions during the entire duration of this project work, without which this work would not have been possible.

I would like to thank my HoD, professors, seniors and my dear friends and specially Mr. Akhilesh Khapre for helping me in this project work. Without their help I wouldn't be able to make it.

Date: 07-05-2012

---

**Rohit Prabhakar**  
**(108CH043)**

## **ABSTRACT**

The project report deals with the analysis of the fluid flow phenomena at different conditions, utilising the CFD simulation software ANSYS. Starting off with the work, a tank of sides 10m is considered. The tank is provided with a cylinder at the centre, with a diameter of 1m. Certain fluid is made to flow through the cylinder, thus rotating the cylinder with the angular velocity ranging from 0rpm to 100rpm. The Reynolds Number is taken into account and is varied from single-digit value 1 to multi-digit value 10,000. On each Reynolds Number value being considered, the cylinder is rotated at different angular velocities ranging from 0rpm to 100rpm. On each of these values, the simulation is done using ANSYS 13.0. Workbench is used for the geometry purpose while Fluent 6.2.16 is utilised for the rest of the simulation purpose. The value of drag coefficient is obtained at all these combinations of Reynolds Number and angular velocity. The tabulation is done and the hence the pattern is observed. Hence a common conclusion is drawn from all the tabulations done. This gives the idea regarding the relationship between angular velocity, Reynolds Number and drag coefficient.

# **CONTENTS**

<b>Chapter</b>		<b>Topic</b>	<b>Page No.</b>
		<b>Abstract</b>	iv
		<b>List of Figures</b>	vii
		<b>List of Tables</b>	ix
		<b>Nomenclature</b>	x
<b>Chapter 1</b>		<b>Introduction</b>	1
	1.1	Newtonian fluid	2
	1.2	Non-Newtonian fluid	2
	1.3	Computational fluid dynamics	2
	1.4	ANSYS, Inc.	3
	1.5	Drag coefficient	3
	1.6	The problem statement	4
<b>Chapter 2</b>		<b>Literature Review</b>	5
	2.1	Methodology	7
	2.2	The literature review for the drag coefficient	8
<b>Chapter 3</b>		<b>Theory</b>	10
	3.1	CFD (Computational Fluid Dynamics)	11
	3.2	Navier-Stokes Equation	11
	3.3	Benefits of CFD	12
	3.4	CFD Process	13
	3.5	Limitations of CFD	13
	3.6	Comparative Study of Experimental, Analytical and Numerical Methods	13

	3.7	Drag Coefficient	15
	3.8	Factors affecting drag coefficient (Cd)	17
	3.9	Reynolds Number	18
<b>Chapter 4</b>		<b>Simulation</b>	20
<b>Chapter 5</b>		<b>Results &amp; Discussion</b>	24
<b>Chapter 6</b>		<b>Conclusion</b>	39
		<b>References</b>	42

## List of Figures

Sl. No.	Figure Name	Page No.
1	Measured drag coefficients	16
2	Shape effects on drag	18
3	Angular Velocity vs Cd graph at Re = 1	25
4	Pressure profile at Re =1 and at 0 rpm	26
5	Pressure profile at Re =1 and at 100 rpm	26
6	Velocity profile at Re =1 and at 0 rpm	26
7	Velocity profile at Re =1 and at 100 rpm	26
8	Angular Velocity vs Cd graph at Re = 2	27
9	Pressure profile at Re = 2 and at 0 rpm	28
10	Pressure profile at Re = 2 and at 100rpm	28
11	Velocity profile at Re = 2 and at 0rpm	28
12	Velocity profile at Re = 2 and at 100rpm	28
13	Angular Velocity vs Cd graph at Re = 5	29
14	Pressure profile at Re = 5 and at 0rpm	30
15	Pressure profile at Re = 5 and at 100rpm	30
16	Velocity profile at Re = 5 and at 0rpm	30
17	Velocity profile at Re = 5 and at 100 rpm	30
18	Angular Velocity vs Cd graph at Re = 10	31
19	Pressure profile at Re = 10 and at 0 rpm	32
20	Pressure profile at Re = 10 and at 100 rpm	32
21	Velocity profile at Re = 10 and at 0 rpm	32
22	Velocity profile at Re = 10 and at 100 rpm	32

23	Angular Velocity vs Cd graph at Re = 100	33
24	Pressure profile at Re =100 and at 0 rpm	34
25	Pressure profile at Re =100 and at 100 rpm	34
26	Velocity profile at Re =100 and at 0 rpm	34
27	Velocity profile at Re =100 and at 100 rpm	34
28	Reynolds Number vs Cd graph at 0 rpm	36
29	Reynolds Number vs Cd graph at 0 rpm (closer look)	36
30	Pressure profile at Re =500 and at 0 rpm	37
31	Pressure profile at Re =1000 and at 0 rpm	37
32	Pressure profile at Re =5000 and at 0 rpm	37
33	Pressure profile at Re =10000 and at 0 rpm	37
34	Velocity profile at Re =500 and at 0 rpm	38
35	Velocity profile at Re =1000 and at 0 rpm	38
36	Velocity profile at Re =5000 and at 0 rpm	38
37	Velocity profile at Re =10000 and at 0 rpm	38



## List of Tables

<b>Table. No</b>	<b>Table Name</b>	<b>Page No.</b>
2.1	Drag coefficients at different flow velocities	9
3.1	Comparison of Experimental, Analytical and Numerical Methods of Solution	14
5.1	Values of Drag Coefficient at Reynolds Number = 1 different angular velocities	25
5.2	Values of Drag Coefficient at Reynolds Number = 2 and different angular velocities	27
5.3	Values of Drag Coefficient at Reynolds Number = 5 and different angular velocities	29
5.4	Values of Drag Coefficient at Reynolds Number = 10 and different angular velocities	31
5.5	Values of Drag Coefficient at Reynolds Number = 100 and different angular velocities	33
5.6	Values of Drag Coefficient at various Reynolds Numbers	35

## Nomenclature:

$\tau$	shear stress exerted by the fluid (drag), Pa
$\mu$	fluid viscosity - a constant of proportionality, Pa·s
$\frac{du}{dy}$	velocity gradient perpendicular to the direction of shear, or equivalently the strain rate, s <sup>-1</sup>
$c_d, c_x, c_w$	drag coefficient, dimensionless
$\mathbf{v}$	flow velocity, m/s
$\rho$	fluid density, kg/m <sup>3</sup>
$p$	Pressure, Pa
$\mathbf{T}$	(deviatoric) stress tensor, N/m <sup>2</sup>
$\mathbf{f}$	body forces (per unit volume) acting on the fluid, N/m <sup>3</sup>
$\nabla$	del operator
$D\mathbf{v}/Dt$	material derivative, m/s <sup>2</sup>
$F_d$	drag force, N
$v$	speed of the object relative to the fluid, m/s
$A$	reference area, m <sup>2</sup>
$D_H$	hydraulic diameter of the pipe, m
$L$	characteristic travelled length, m
$Q$	volumetric flow rate, m <sup>3</sup> /s
$\mu$	dynamic viscosity of the fluid, Pa·s or N·s/m <sup>2</sup> or kg/(m·s)

# CHAPTER 1

## INTRODUCTION

## **INTRODUCTION:**

**1.1 Newtonian fluid** :- A fluid whose stress versus strain rate curve, when plotted, passes through the origin and is linear is known as a **Newtonian fluid**. It is named after Sir Isaac Newton. Mathematically, it can be defined as,

$$\tau = \mu \frac{du}{dy}$$

Where,

$\tau$  is the shear stress exerted by the fluid (drag) [Pa]

$\mu$  is the fluid viscosity - a constant of proportionality [Pa.s]

$\frac{du}{dy}$  is the velocity gradient perpendicular to the direction of shear, or equivalently the strain rate [ $s^{-1}$ ] [1]

**1.2 Non-Newtonian fluid** :- A fluid whose flow properties differ in any way from those of Newtonian fluids. Most commonly the viscosity (resistance to deformation or other forces) of non-Newtonian fluids is not independent of shear rate or shear rate history. In case of such a fluid, the plot between the shear stress and the shear rate is different, and can even be time-dependent. Hence, a constant coefficient of viscosity cannot be defined.

Therefore, although the concept of viscosity is commonly used in fluid mechanics to characterize the shear properties of a fluid, it can be inadequate to describe non-Newtonian fluids. They are best studied through several other rheological properties which relate stress and strain rate tensors under many different flow conditions, such as oscillatory shear, or extensional flow which are measured using different devices or rheometers. The properties can be studied better with the use of tensor-valued constitutive equations. These equations are common in the field of continuum mechanics. Examples can be listed as oobleck, flubber, chilled caramel topping, silly putty, ketchup etc.[2]

**1.3 Computational fluid dynamics** :- It is usually abbreviated as **CFD** and is defined as a branch of fluid mechanics that solves and analyzes fluid flow problems, using numerical methods and algorithms. In order to perform the calculations required to simulate the fluid-surface interaction, defined by boundary conditions, computers need to be employed. Advantage in employing high-speed supercomputers is that it provides better solutions.

The fundamental basis of almost each and every CFD problems is linked with the Navier–Stokes equations. These define almost any single-phase fluid flow. The Navier–

Stokes equations can be simplified by removing certain parameters, that describe viscosity and leads to Euler equations. Further simplification can be done by removing the parameters that describe vorticity and hence yields the full potential equations. After the simplifications are done, these equations can be then linearized so as to obtain the linearized potential equations.[3]

**1.4 ANSYS, Inc. :-** It is an engineering simulation software (computer-aided engineering, or simply CAE in short) developer that is headquartered in Canonsburg, Pennsylvania, United States. The company was founded in 1970 by Dr. John A. Swanson and was originally named **Swanson Analysis Systems, Inc.**

ANSYS offers a comprehensive range of engineering simulation solution sets providing access to virtually any field of engineering simulation that a design process requires :-

- **Simulation Technology** Structural Mechanics Multiphysics Fluid Dynamics Explicit Dynamics Electromagnetics
- **Workflow Technology** ANSYS Workbench Platform High-Performance Computing Geometry Interfaces Simulation Process & Data Management [4]

The present project work utilizes ANSYS 13.0 where Workbench is used for the geometry purpose while the further simulation is done using Fluent 6.2.16. The velocity and pressure profiles are observed at certain angular velocity of the cylinder and certain velocity of the flowing liquid. The drag coefficient is hence obtained and further verified with the standard results.

**1.5 Drag coefficient :-** In context of the fluid dynamics, the **drag coefficient** (commonly denoted as:  $c_d$ ,  $c_x$  or  $c_w$ ) is defined as a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment such as air or water. It is used in the drag equation, where a lower drag coefficient indicates the object will have less aerodynamic or hydrodynamic drag. The drag coefficient is always associated with a particular surface area. [5]

The drag characteristics of objects in case of fluid flow is essential to understand, specially for engineering design aspects. Such knowledge can be employed to reduce the drag on automobiles, aircrafts, and buildings. The phenomena of drag can be simulated (using softwares like ANSYS, Fluent etc.) and measured by recreating water flow over the object. The

behavior of fluid flow around an object can be studied by combining the theories such as conservation of momentum and continuity, with the experimental data.

**1.6 The problem statement** - The present project work consists of a cubical container of sides 10m each. On the centre of the container is kept a cylinder of diameter 1m. The fluid is made to flow in the negative Y direction at a velocity of 1m/s. The effect of gravity is not being considered in this case. The density of the flowing fluid is assumed to be 1kg/m<sup>3</sup>. The Reynolds Number of the fluid is being varied from 1 to 10,000. The viscosity is accordingly calculated using the standard formula for Reynolds Number. On each value of Reynolds Number being considered, the value of angular velocity is varied from 0rpm to 100rpm. The geometry of the problem is created using Workbench, followed by meshing. The rest of work which includes providing initial and boundary conditions etc is done in Fluent. Thus the drag co-efficient on each value being considered, is calculated using ANSYS. Tabulation is done and the values of the drag co-efficient are observed to get an idea about the relation among the Reynolds Number, angular velocity and the drag co-efficient.

# CHAPTER 2

## **LITERATURE REVIEW**

# LITERATURE REVIEW

Historically, methods were first developed for the purpose of solving the Linearized Potential equations. Using the conformal transformations of flow about a cylinder to the flow about an airfoil, the two-dimensional methods were developed in the 1930s.[6] The development of three-dimensional methods was paced by the computer power available then. In 1967, the first paper on a practical three-dimensional method to solve the linearized potential equations was published by A.M.O. Smith and John Hess of Douglas Aircraft.[7] The description of the first lifting Panel Code (A230) was done in 1968, in a paper written by Gary Saaris and Paul Rubbert of Boeing Aircraft. It was during this period that more advanced three-dimensional Panel Codes were developed at Boeing (PANAIR, A502), Lockheed (Quadpan), McDonnell Aircraft (MACAERO), NASA (PMARC), Douglas (HESS) and Analytical Methods (WBAERO, USAERO and VSAERO). Some (PANAIR, HESS and MACAERO) were higher order codes, utilising higher order distribution of surface singularities, whereas others (Quadpan, PMARC, USAERO and VSAERO), on each surface panel, used single singularities. A great benefit of the lower order codes was that they ran much faster on the computers of the time.

For the purpose of airfoil analysis and design, a number of Panel Codes have been developed in the two-dimensional realm. In order to model the viscous effects, the codes typically includes a boundary layer analysis. This was soon followed by XFOIL code, developed by Mark Drela, an MIT Professor. Both XFOIL and PROFIL incorporate 2-D panel codes, with coupled boundary layer codes for airfoil analysis work. While XFOIL has both an inverse panel and a conformal transformation method for airfoil design, on the other hand, PROFIL uses only a conformal transformation method for inverse airfoil design. Both codes are used.

An intermediate step between Panel Codes and Full Potential codes were the codes that utilized the Transonic Small Disturbance equations. In particular, the three-dimensional WIBCO code, developed in the early 1980s by Charlie Boppe of Grumman Aircraft has found heavy use.

When a further growth of Program H was developed at Grumman Aerospace as Grumfoil, the developers Bob Melnik and his group, turned to Full Potential codes. Later, Antony Jameson, originally at Grumman Aircraft and the Courant Institute of NYU, and David Caughey, worked together and developed the important 3-D Full Potential code



FLO22 in 1975. This led to the development of many Full Potential codes, that further culminated in Boeing's Tranair (A633) code, still under heavy use.

Later on, emerged the **Euler** equations, with a promise to produce better and accurate solutions for the problems related to transonic flows. The methodology used by Jameson in his three-dimensional FLO57 code (1981) was used by others to produce programs, similar to IAI/Analytical Methods' MGAERO program and Lockheed's TEAM program. MGAERO is a structured cartesian mesh code and that accounts for its uniqueness. On the other hand, most other such codes use structured body-fitted grids (exceptions being NASA's highly successful CART3D code, Georgia Tech's NASCART-GT and Lockheed's SPLITFLOW code).[8] Antony Jameson also, with the use of unstructured tetrahedral grids, developed the three-dimensional AIRPLANE code in 1985.

However, the ultimate target of the developers were the **Navier–Stokes equations**. In this process, first emerged the 2-D codes like NASA Ames' ARC2D code. Further, a number of 3-D codes were developed. The three successful NASA contributions are listed as :-

- i. ARC3D
- ii. OVERFLOW
- iii. CFL3D

This further lead to the development of numerous commercial packages.

**2.1 Methodology** :- Out of all the approaches listed above, almost all of them apply the same basic procedure. During pre-processing :-

- The structures used in the problem are drawn and the measurements, geometry, physical bounds are defined.
- The total volume of the fluid is undergone through meshing procedure and hence is divided into discrete cells, termed as the mesh. This mesh thus generated can be either uniform or non uniform.
- The next step called Setup is then processed where the physical modeling is defined.
- Boundary conditions are defined. This includes specification of the fluid properties and behaviour at the boundaries of the problem. In case of transient problems, the definition of initial conditions is must.

- The simulation is then begun and hence the equations are processed iteratively as a steady-state or transient.
- Finally a postprocessor is used for the analysis and visualization of the resulting solution.

**ANSYS, Inc.** being used here for simulation, is an engineering simulation software (computer-aided engineering, or CAE) developer with its headquarters located south of Pittsburgh in Canonsburg, Pennsylvania, United States.

Dr. John A. Swanson, in 1970, led to the foundation of the company. Initially, it was named as **Swanson Analysis Systems, Inc.** or SASI (after the name of its founder). The primary purpose of this company was to develop and market finite element analysis software for structural physics that could simulate static (stationary), dynamic (moving) and heat transfer (thermal) problems. SASI developed its business in parallel with the growth in engineering needs and the computer technology. It grew by 10 percent to 20 percent each year, and finally was sold in 1994. The leading software of SASI, named ANSYS®, was taken as the company's flagship product by the new owners. Hence ANSYS, Inc. was designated as the new name of the company. A number of companies since 2000, have been acquired by the company. The list includes Harvard Thermal, CFX (2003), ICEM CFD Engineering, CADOE, Fluent Inc. (2006), Century Dynamics, Ansoft Corporation (2008) and Apache Design Solutions (2011). ANSYS was listed on the NASDAQ stock exchange in 1996. In late 2011, Investor's Business Daily ranked ANSYS as one of only six technology businesses worldwide to receive the highest possible score on its SmartSelect Composite Ratings.[9] ANSYS has been recognized as a strong performer by a number of other sources as well.[10] The organization reinvests 15 percent of its revenues each year into research to continually refine the software.[11]

**2.2 The literature review for the drag coefficient** :- The literature review can be obtained from an experimental work performed in 2003 by Giancarlo Bruschi, Tomoko Nishioka, Kevin Tsang and Rick Wang. The flow considered by them was in positive X direction. The values of the drag co-efficient obtained by them at different flow velocities was given as follows :-

**Table 2.1 – Drag coefficients at different flow velocities :-**

Flow Velocity (m/s)	Cd	Cd	Cd
0.9	1.21	0.91	1.06
1.4	1.205	1.11	1.34
1.75	1.183	1.18	1.24

The different values of drag co-efficient have been calculated experimentally at three different conditions, specified in much detail in the report submitted by them. Thus, the idea about the values of drag coefficient at different instances was obtained and was found to be around 1. [12]

# CHAPTER 3

## **THEORY**

### 3.1 CFD (Computational Fluid Dynamics)

CFD is a technology that enables us to study the dynamics of things that flow. Using CFD we can develop a computational model of the system or device that we want to study. Fluid flow physics is applied along with some chemistry and the software will output the fluid dynamics and the related physical phenomena. With the advent of CFD, one has the power to simulate the flow of gases and liquids, heat and mass transfer moving bodies, multiphase physics, chemical reaction, fluid structure interaction and acoustics through computer modeling. A virtual prototype of the system can be built using CFD (Fluent Inc, 2001). Thus CFD can be applied for predicting the fluid flow associated with the complications of simultaneous flow of heat, mass transfer, phase change, chemical reaction, etc. using computers. CFD has now become an integral part of the engineering design and analysis. Engineers can make optimal use of the CFD tools to simulate fluid flow and heat transfer phenomena in a system model and can even predict the system performance before actually manufacturing it. [14]

### 3.2 Navier-Stokes Equation :-

It is named after Claude-Louis Navier and George Gabriel Stokes and describes the motion of fluid substances. It is also a fundamental equation being used by ANSYS and even in the present project work being carried out. These equations arise from applying second law of Newton to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term.

The derivation of the Navier–Stokes equations begins with an application of second law of Newton i.e. conservation of momentum (often alongside mass and energy conservation) being written for an arbitrary portion of the fluid. In an inertial frame of reference, the general form of the equations of fluid motion is :- [15]

**Navier–Stokes equations (general)**

$$\rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \nabla \cdot \mathbb{T} + \mathbf{f},$$

Where

$\mathbf{v}$  is the flow velocity,

$\rho$  is the fluid density,

$p$  is the pressure,

$\mathbb{T}$  is the (deviatoric) stress tensor,

$\mathbf{f}$  represents body forces (per unit volume) acting on the fluid,

$\nabla$  stands for the del operator.

This equation is often written using the material derivative, denoted as  $D\mathbf{v}/Dt$ , making it more apparent that this is a statement of second law of Newton :-

$$\rho \frac{D\mathbf{v}}{Dt} = -\nabla p + \nabla \cdot \mathbb{T} + \mathbf{f}.$$

The left side of the equation describes acceleration, and may be composed of time dependent or convective effects (also the effects of non-inertial coordinates if present). The right side of the equation is in effect a summation of body forces (such as gravity) and divergence of stress (pressure and shear stress). [16]

### 3.3 Benefits of CFD

- ✚ Insight- if there is a device or system design which is difficult to analyze or test through experimentation, CFD analysis enables us to virtually sneak inside the design and see how it performs. CFD gives a deep perception into the designs. There are many occurrences that we can witness through CFD which wouldn't be visible through any other means.
- ✚ Foresight- under a given set of circumstances, we can envisage through the CFD software what will happen. In a short time we can predict how the design will perform and test many variants until we arrive at an ideal result.
- ✚ Efficiency- the foresight we gain helps us to design better to achieve good results. CFD is a device for compressing the design and development cycle allowing for rapid prototyping.

Advantages of CFD can be summarized as :- [17]

- ✚ The effect of various parameters and variables on the behavior of the system can be studied instantaneously since the speed of computing is very high. To study the same in an experimental setup is not only difficult and tedious but also sometimes may be impossible.

- ✚ In terms of cost factor, CFD analysis will be much cheaper than setting up experiments or building sample model of physical systems.
- ✚ Numerical modeling is flexible in nature. Problems with different level of complexity can be simulated.
- ✚ It allows models and physical understanding of the problem to be improved, very much similar to conducting experiments.
- ✚ In some cases it may be the only practicable ancillary for experiments.

### 3.4 CFD Process

The steps underlying the CFD process are as follows:

- ✚ Geometry of the problem is defined.
- ✚ Volume occupied by fluid is divided into discrete cells.
- ✚ Physical modeling is well-defined.
- ✚ Boundary conditions are defined which involves specifying the fluid behavior and properties at the boundaries.
- ✚ Equations are solved iteratively as steady state or transient state.
- ✚ Analysis and visualization of resulting solution is carried out.

### 3.5 Limitations of CFD


Even if there are many advantages of CFD, there are few shortcomings of it as follows :-  
[18]


- ✚ CFD solutions rely upon physical models of real world processes.
- ✚ Solving equations on a computer invariably introduces numerical errors.
- ✚ Truncation errors due to approximation in the numerical models.
- ✚ Round-off errors due to finite word size available on the computer.
- ✚ The accuracy of the CFD solution depends heavily upon the initial or boundary conditions provided to numerical model.

### 3.6 Comparative Study of Experimental, Analytical and Numerical Methods

✚ **Experimental Method** - Experimental methods are used to obtain consistent information about physical processes which are not clearly understood. It is the most

realistic approach for problem solving. It may involve full scale, small scale or blown up scale model. However disadvantages are high cost, measurement difficulties and probe errors.

 **Analytical Method** - These methods are used to obtain solution of mathematical model which consists of a set of differential equations that represent a physical process within the limit of conventions made. The systematic solution often contain infinite series, special functions etc. and hence their numerical evaluation becomes difficult to handle.

 **Numerical method** – Numerical prediction works on the results of the mathematical model. The solution is obtained for variables at distinct grid points within the computational field. It provides for greater handling of complex geometry and non-linearity in governing equations or boundary conditions. The kind of ease provided by numerical methods makes it the powerful and widely applicable. The above said discussion is represented in tabular form in table 3.1

**Table 3.1. Comparison of Experimental, Analytical and Numerical Methods of Solution [17]**

Name of the Method	Advantages	Disadvantages
1. Experimental	Capable of being most realistic	<ul style="list-style-type: none"> <li>• Equipment required</li> <li>• Scaling problem</li> <li>• Measurement difficulties</li> <li>• Probe errors</li> <li>• High operating costs</li> </ul>
2. Analytical	Clean, general information which is usually in formula form	<ul style="list-style-type: none"> <li>• Restricted to simple geometry and physics</li> <li>• Usually restricted to linear problems</li> <li>• Cumbersome results- difficult to compute</li> </ul>



3. Numerical	<p>No restriction to linearity.</p> <p>Ability to handle irregular geometry and complicate physics.</p> <p>Low cost and high speed of computation.</p>	<ul style="list-style-type: none"> <li>• Truncation and round-off errors</li> <li>• Boundary condition problems</li> </ul>
--------------	--	--

An assessment of advantages and disadvantages of numerical methods vis-à-vis analytical and experimental method shows that even though the Numerical Method has few shortcomings but it has many advantages associated with it and is hence suited. [19]

### 3.7 Drag Coefficient :-

The drag coefficient  $c_d$  is defined as:

$$c_d = \frac{2F_d}{\rho v^2 A},$$

where:

$F_d$  is the drag force, which is by definition the force component in the direction of the flow velocity,

$\rho$  is the mass density of the fluid,

$v$  is the speed of the object relative to the fluid and










$A$  is the reference area.

The reference area depends on what type of drag coefficient is being measured. For automobiles and many other objects, the reference area is the projected frontal area of the vehicle. This may not necessarily be the cross sectional area of the vehicle, depending on where the cross section is taken. For example, for a sphere  $A = \pi r^2$  (note this is not the surface area  $= 4\pi r^2$ ).

For airfoils, the reference area is the planform area. Since this tends to be a rather large area compared to the projected frontal area, the resulting drag coefficients tend to be low: much lower than for a car with the same drag and frontal area, and at the same speed.

Airships and some bodies of revolution use the volumetric drag coefficient, in which the reference area is the square of the cube root of the airship volume. Submerged streamlined bodies use the wetted surface area.

Two objects having the same reference area moving at the same speed through a fluid will experience a drag force proportional to their respective drag coefficients. Coefficients for unstreamlined objects can be 1 or more, for streamlined objects much less. [20]

Shape		Drag Coefficient
Sphere	→ 	0.47
Halfsphere	→ 	0.42
Cone	→ 	0.50
Cube	→ 	1.05
Angled Cube	→ 	0.80
Long Cylinder	→ 	0.82
Short Cylinder	→ 	1.15
Streamlined Body	→ 	0.04
Streamlined Halfbody	→ 	0.09

Measured Drag Coefficients

**Fig. 1**

However, it must be noted that  $C_d$  is not an absolute constant for any given body shape. Its value changes with the speed of the airflow and even more with the change in Reynolds number. The smooth sphere considered in the above figure, for example, has a  $C_d$  that differs from high values in case of laminar flow to values as low as 0.47 for turbulent flow (the latter being considered here).

### 3.8 Factors affecting drag coefficient ( $C_d$ ) :-

The factors which affect the drag coefficient value can be grouped into following factors :-

1. **Those associated with the object [Shape and size]** - Geometry has a large effect on the amount of drag generated by an object. As with lift, the drag depends linearly on the size of the object moving through the fluid. The cross-sectional shape of an object determines the **form drag** created by the pressure variation around the object. The amount of drag depends on the surface roughness of the object; a smooth, waxed surface produces less drag than a roughened surface. This effect is called **skin friction** and is usually included in the measured drag coefficient of the object.



## Size Effects on Drag

Glenn  
Research  
Center

**Drag is directly related to reference area**

$$D = \text{Constant} \times A_{\text{ref}}$$

**Double the Area --> Double the Drag** [21]

2. **Those associated with the motion of the object through the fluid [velocity, inclination to flow]** - Drag depends on the velocity of the fluid. Like lift, drag actually varies with the square of the relative velocity between the object and the fluid. The inclination of the object to the flow also affects the amount of drag generated by a given shaped object. The motion of the object through the fluid also causes boundary layers to form on the object. A boundary layer is a region of very low speed flow near the surface which contributes to the **skin friction**.
3. **Those associated with the fluid itself [Mass, viscosity, compressibility]** - Drag depends directly on the mass of the flow going past the body. The drag also depends in a complex way on two other properties of the fluid: its viscosity and its

compressibility. We can gather all of this information on the factors that affect drag into a single mathematical equation called the Drag Equation. With the drag equation we can predict how much drag force is generated by a given body moving at a given speed through a given fluid.[22]



## Shape Effects on Drag

Glenn  
Research  
Center

The shape of an object has a very great effect on the amount of drag.

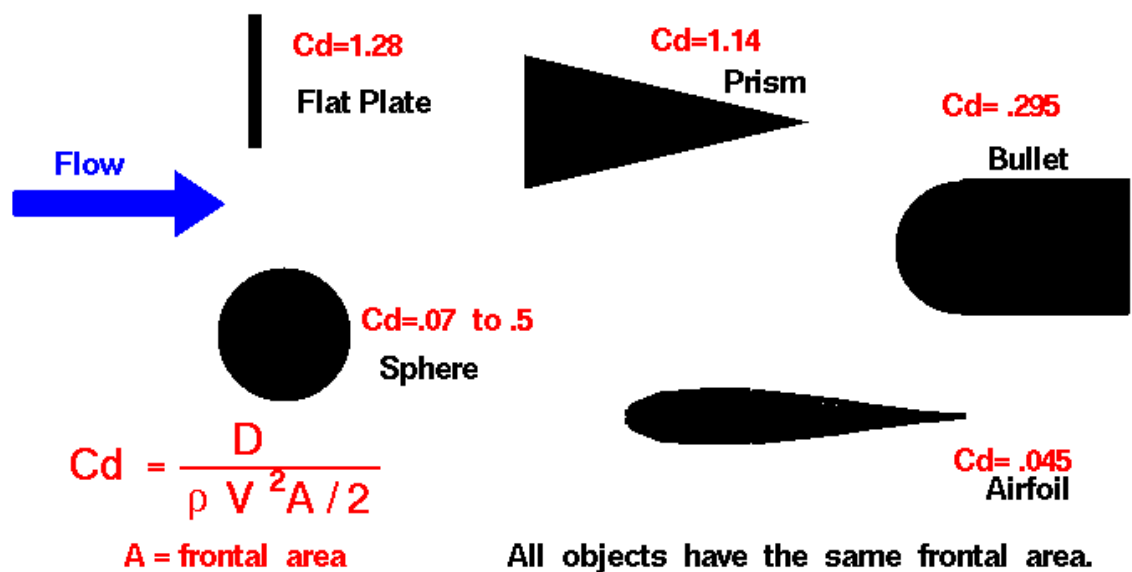


Fig. 2

To summarize, for any object immersed in a fluid, the mechanical forces are transmitted at every point on the surface of the body. The forces are transmitted through the pressure, which acts perpendicular to the surface. The net force can be found by integrating (or summing) the pressure times the area around the entire surface. For a moving flow, the pressure will vary from point to point because the velocity varies from point to point. [23]

### 3.9 Reynolds Number :-

In context of fluid mechanics, the **Reynolds number (Re)** is defined as a dimensionless number that gives a measure of the ratio of inertial forces to viscous forces and consequently quantifies the relative importance of these two types of forces for given flow conditions. The concept of this number was introduced by George Gabriel Stokes in

1851, [24] but the Reynolds number is named after Osborne Reynolds (1842–1912), who popularized its use in 1883. [25][26]

For flow in a pipe or tube, the Reynolds number is generally defined as :- [27]

$$\text{Re} = \frac{\rho v D_H}{\mu} = \frac{v D_H}{\nu} = \frac{Q D_H}{\nu A}$$

where:

- $D_H$  is the hydraulic diameter of the pipe
- its characteristic travelled length,  $L$ , (m).
- $Q$  is the volumetric flow rate ( $\text{m}^3/\text{s}$ ).
- $A$  is the pipe *cross-sectional* area ( $\text{m}^2$ ).
- $v$  is the mean velocity of the object relative to the fluid (SI units:  $\text{m/s}$ ).
- $\mu$  is the dynamic viscosity of the fluid ( $\text{Pa}\cdot\text{s}$  or  $\text{N}\cdot\text{s}/\text{m}^2$  or  $\text{kg}/(\text{m}\cdot\text{s})$ ).
- $\nu$  is the kinematic viscosity ( $\nu = \mu/\rho$ ) ( $\text{m}^2/\text{s}$ ).
- $\rho$  is the density of the fluid ( $\text{kg}/\text{m}^3$ ).

# CHAPTER 4

## **SIMULATION**

The step by step solution consists of 7 major steps discussed as follows :-

### **Step 1** - Pre-analysis and start-up

- ▶ Start workbench
- ▶ Drag and drop “Fluid Flow (FLUENT)” on “Project Schematic”
- ▶ Select “2D”.

### **Step 2 – Geometry**

For creating the surfaces :-

- ▶ Double click on geometry.
- ▶ XY Plane → Sketching → Rectangle → +Z → Draw rectangle
- ▶ Dimensions → Select edges → give measurements [10m X 10m]
- ▶ Concept → Surface from sketches → Select edges → Apply → Generate
- ▶ XY Plane → Sketching → Circle → Draw circle at centre of rectangle.
- ▶ Dimensions → Select diameter → Enter diameter of cylinder [1m]
- ▶ Concept → Surface from sketches → Select circle → Apply → Generate.

Now for creating Named Selection :-

- ▶ Select Edge Selection → Select the bottom edge → Right click
- ▶ Select “Named Selection” → Name it “Bottom” → Apply → Generate
- ▶ Select the right edge → Right click
- ▶ Select “Named Selection” → Name it “Right” → Apply → Generate
- ▶ Select the top edge → Right click
- ▶ Select “Named Selection” → Name it “Top” → Apply → Generate
- ▶ Select the left edge → Right click
- ▶ Select “Named Selection” → Name it “Left” → Apply → Generate
- ▶ Select the cylinder’s edge → Right click

- ▶ Select “Named Selection” → Name it “Cylinder” → Apply → Generate
- ▶ Close geometry.

### **Step 3 – Mesh**

- ▶ Double click on Mesh
- ▶ Sizing → Relevance Centre → Fine
- ▶ Generate mesh
- ▶ Save → Exit → Update.

### **Step 4 - Setup (Physics) :-**

- Double click on Setup → Select "Double Precision".
- Materials → Fluid → Create/Edit...  
(here, set Density =  $1\text{kg/m}^3$  and Viscosity =  $0.01\text{ kg/ms}$  )
- Boundary Conditions → wall-surface\_body → Edit..  
(here, set Wall Motion to Moving Wall  
Motion to Absolute)
- Rotational → Speed =  $0\text{rpm}$  → X=5, Y=5
- Wall-surface\_body.1 → Type → pressure-outlet.
- Wall-surface\_body.3 → Edit → Velocity specification → Components  
→ Reference frame → Absolute  
→ Y velocity to  $-1\text{m/s}$
- Reference Values → Area =  $10\text{m}^2$   
Length =  $10\text{m}$   
Velocity =  $1\text{m/s}$   
Viscosity =  $0.01\text{kg/ms}$

### **Step 5 – Solution :-**

- ▶ Solution Methods → Momentum → First Order Upwind.
- ▶ Solution Initialization → Initialize.
- ▶ Monitors → Residuals → Edit... → Change all 3 values to  $1\text{e-}6$  → OK.
- ▶ Run Calculation → Number of Iterations ( $10000$ , say) → Calculate.



- ▶ Save project.
- ▶ The value of Viscosity (in Materials) is then varied as per the variation of Reynolds Number from 1 to 10000. On the values of Reynolds Number being considered, the angular velocity is varied from 0rpm to 100rpm.
- ▶ The drag coefficients thus obtained are further tabulated to give the results.

The remaining two steps in the simulation procedure consists of :-

- i. Results
- ii. Verification and Validation

These steps are further discussed in detail and the conclusion is drawn from them.

# CHAPTER 5

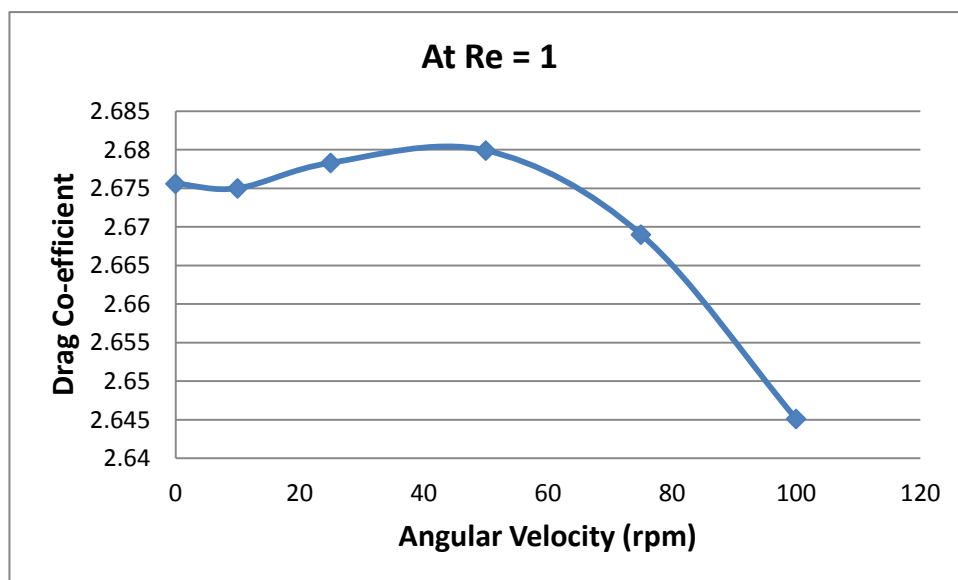
## **RESULTS AND DISCUSSION**

Firstly, the value of the Reynolds Number is kept constant at 1 and the value of angular velocity is varied from 0 rpm to 100 rpm. The drag co-efficient at each angular velocity being considered, is obtained by simulation. Hence, the tabulation gives :-

**Table 5.1 – Values of Drag Coefficient at Reynolds Number = 1 and different angular velocities :-**

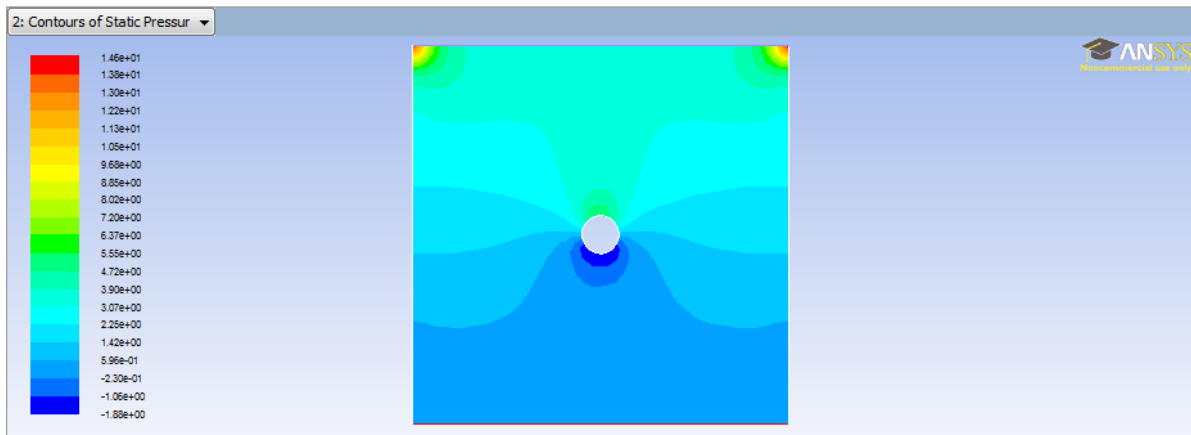
Angular velocity (rpm)	Drag coefficient (Cd)
0	2.6756
10	2.6750
25	2.6783
50	2.6799
75	2.6690
100	2.6451

The graph obtained for these values is plotted as follows :-

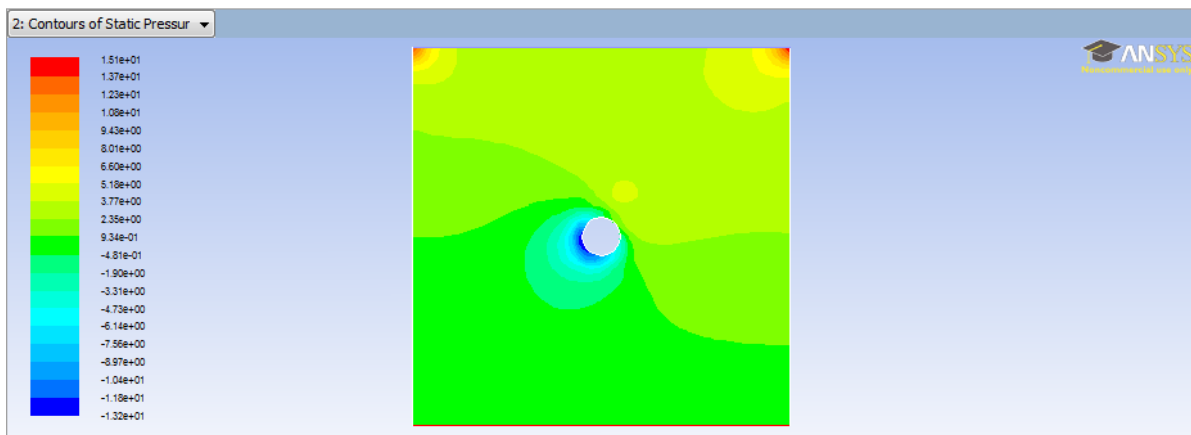


**Fig. 3 Angular Velocity vs Cd graph at Re = 1**

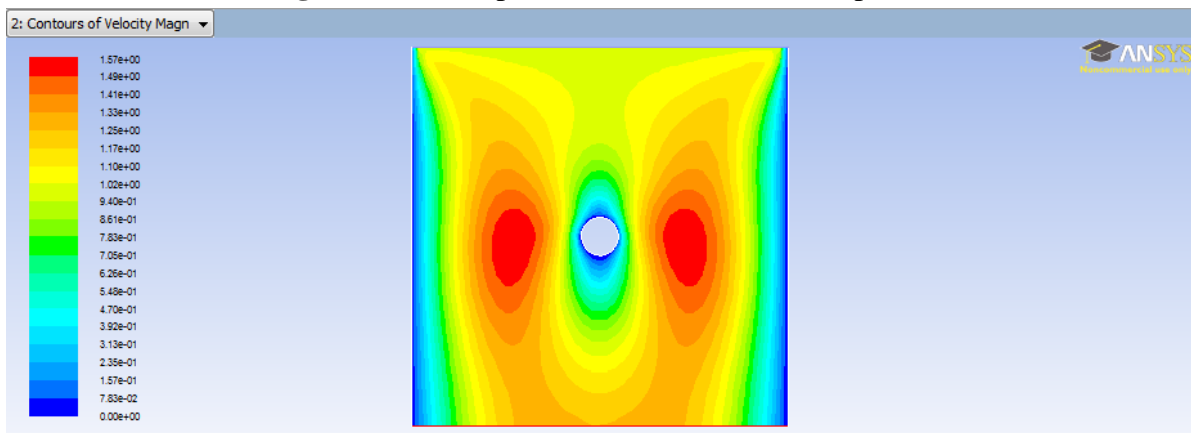
Now, the pressure profiles and the velocity profiles at 0 rpm and 100 rpm are observed to be as follows :-



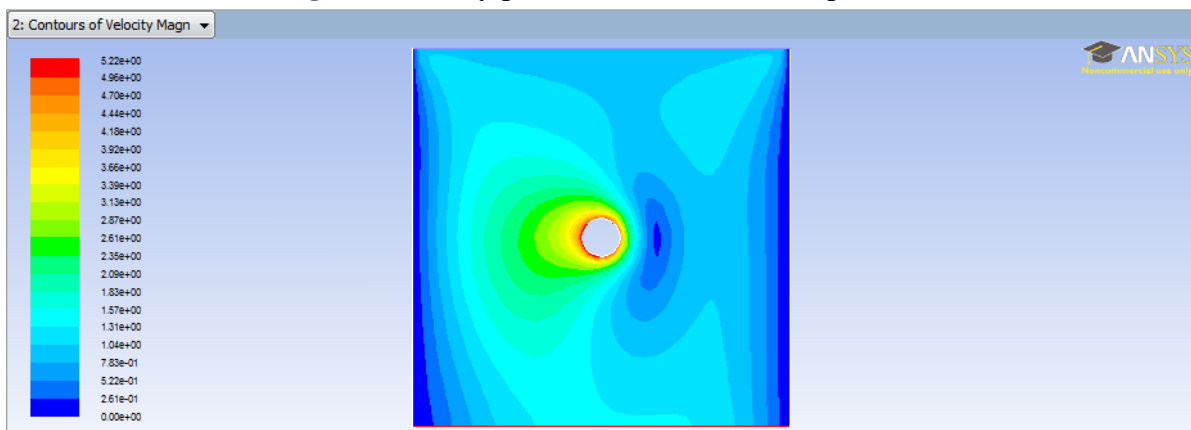
**Fig. 4** - Pressure profile at  $Re = 1$  and at 0rpm



**Fig. 5** - Pressure profile at  $Re = 1$  and at 100rpm



**Fig. 6** - Velocity profile at  $Re = 1$  and at 0rpm



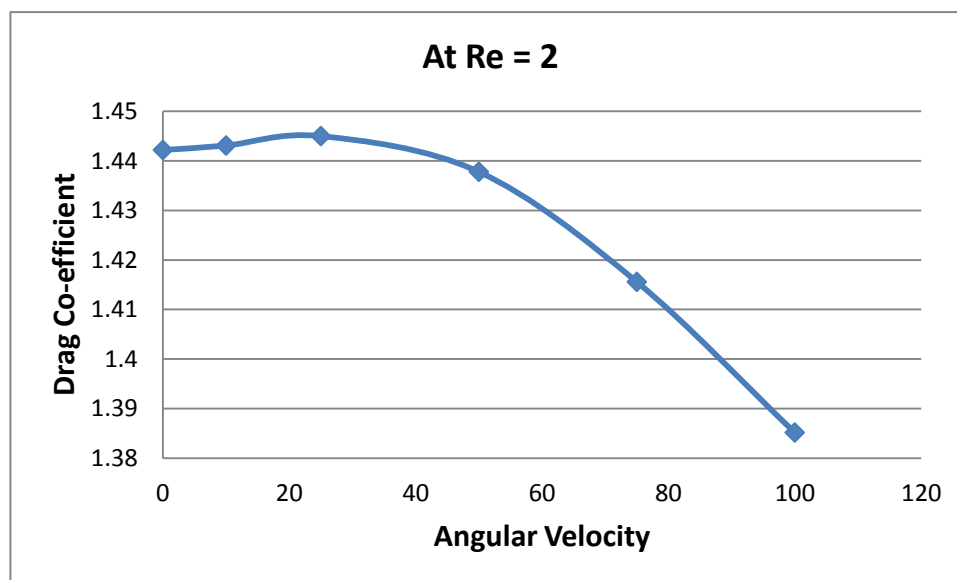
**Fig. 7** - Velocity profile at  $Re = 1$  and at 100rpm

Now the value of Reynolds Number is changed to 2 and the values of drag co-efficient, with angular velocity varying from 0 rpm to 100 rpm is tabulated as follows :-

**Table 5.2 - Values of Drag Coefficient at Reynolds Number = 2 and different angular velocities :-**

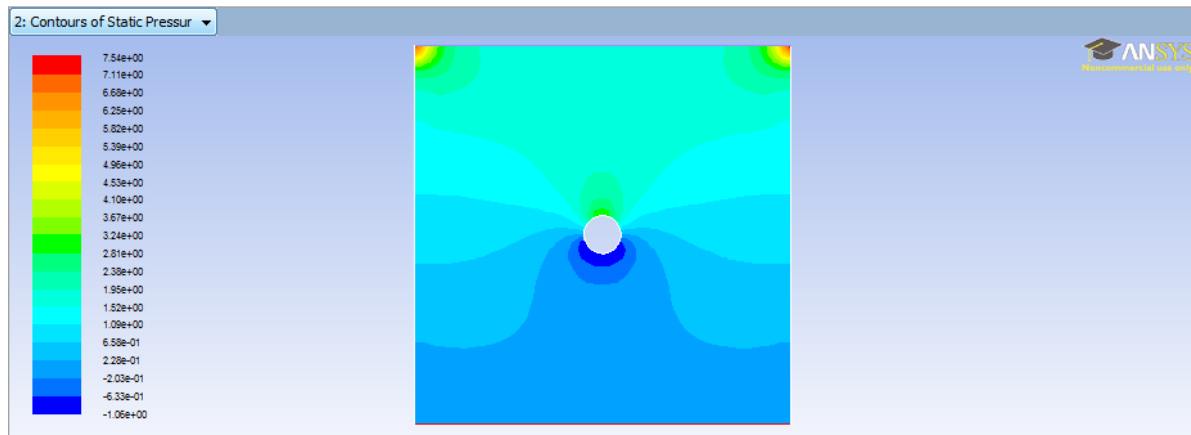
Angular velocity (rpm)	Drag coefficient (Cd)
0	1.4422
10	1.4431
25	1.4450
50	1.4378
75	1.4156
100	1.3852

The graph obtained between the angular velocity and drag co-efficient is plotted as follows :-

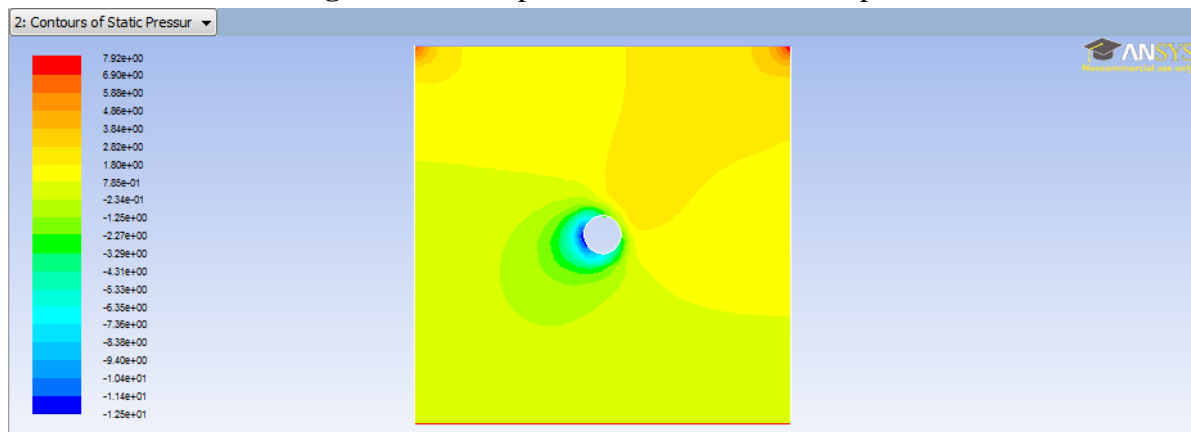


**Fig. 8 Angular Velocity vs Cd graph at Re = 2**

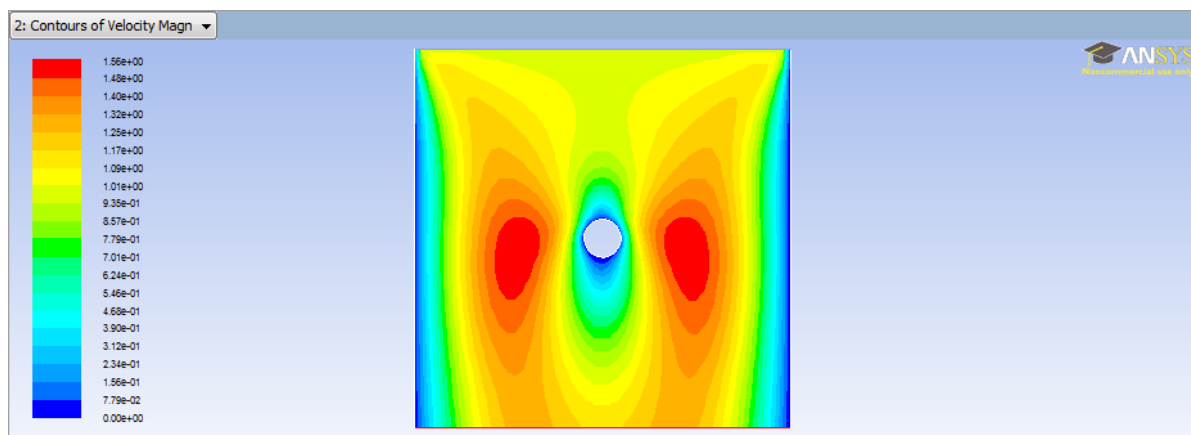
Now, the pressure profiles and the velocity profiles at 0 rpm and 100 rpm are observed to be as follows :-



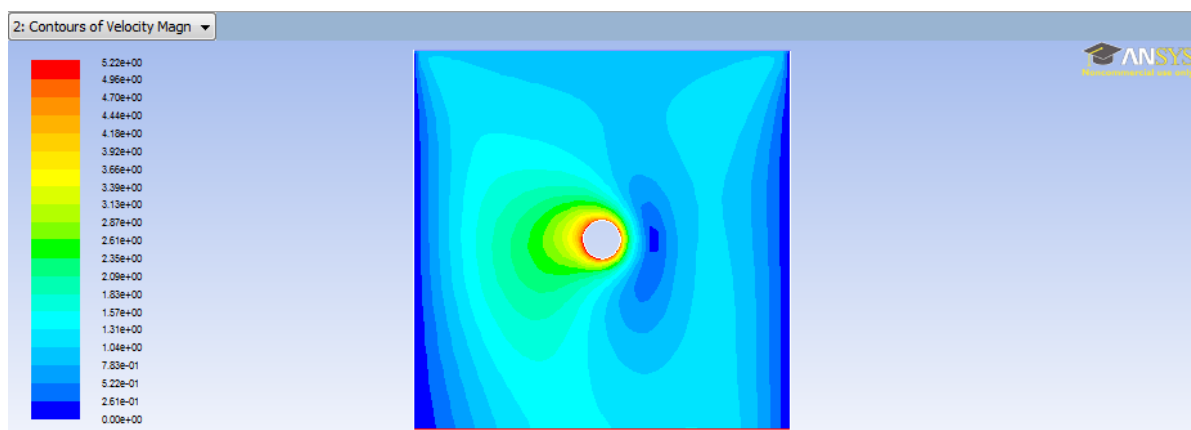
**Fig. 9** - Pressure profile at  $Re = 2$  and at 0 rpm



**Fig. 10** - Pressure profile at  $Re = 2$  and at 100rpm



**Fig. 11** - Velocity profile at  $Re = 2$  and at 0rpm

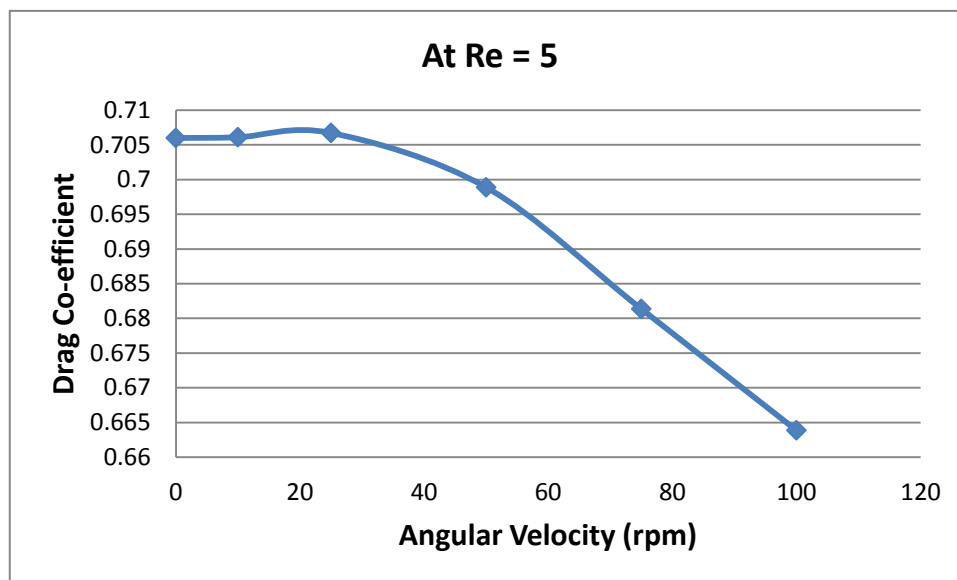


**Fig. 12** - Velocity profile at  $Re = 2$  and at 100rpm

Now, the value of Reynolds Number is changed to 5. The value of angular velocity is again varied from 0 rpm to 100 rpm and the values of drag co-efficient is listed as below :-

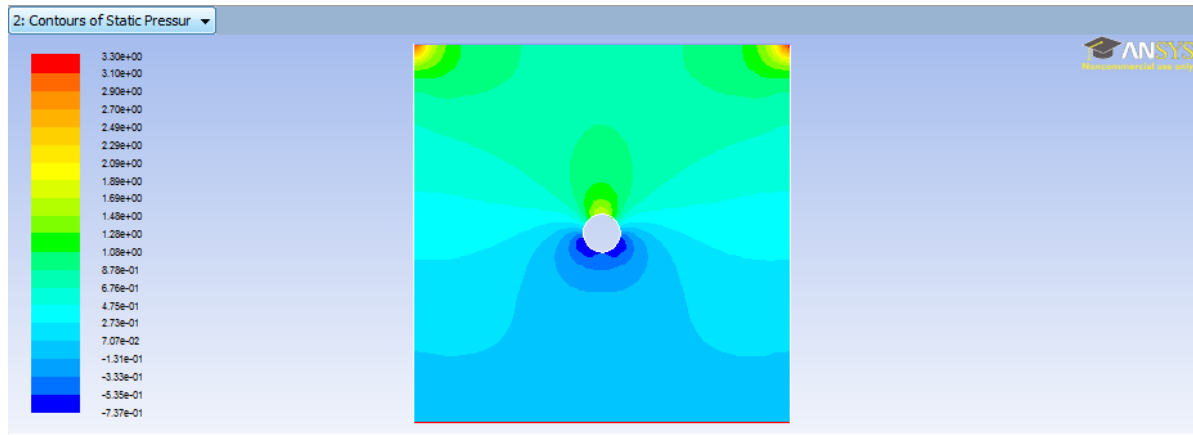
**Table 5.3 - Values of Drag Coefficient at Reynolds Number = 5 and different angular velocities :-**

Angular velocity (rpm)	Drag coefficient (Cd)
0	0.70599
10	0.70610
25	0.70672
50	0.69888
75	0.68136
100	0.66387

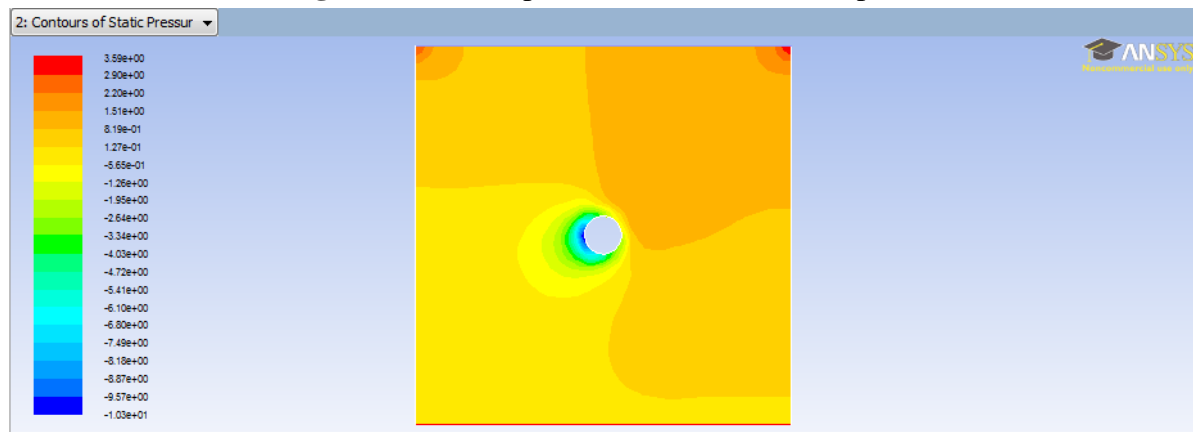


**Fig. 13 - Angular Velocity vs Cd graph at Re = 5**

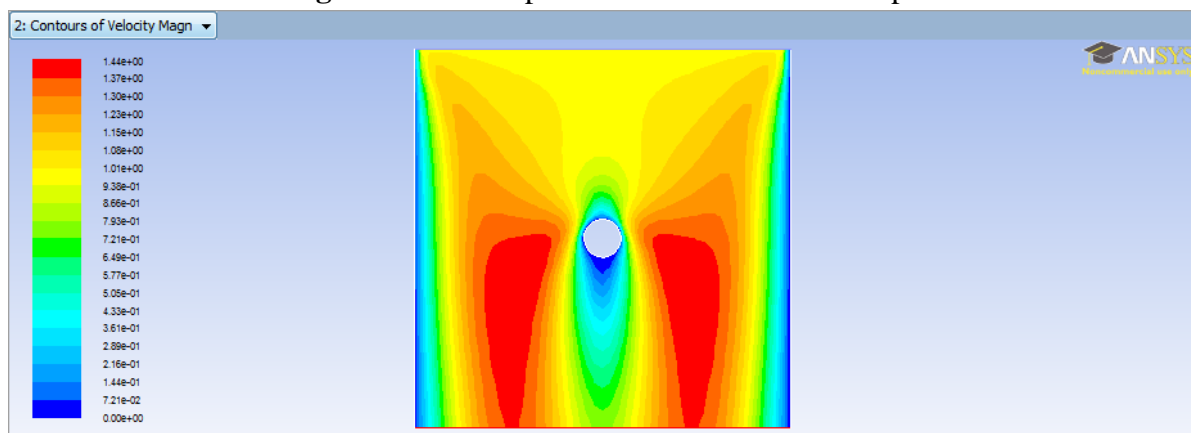
Now, the pressure profiles and the velocity profiles at 0 rpm and 100 rpm are observed to be as follows :-



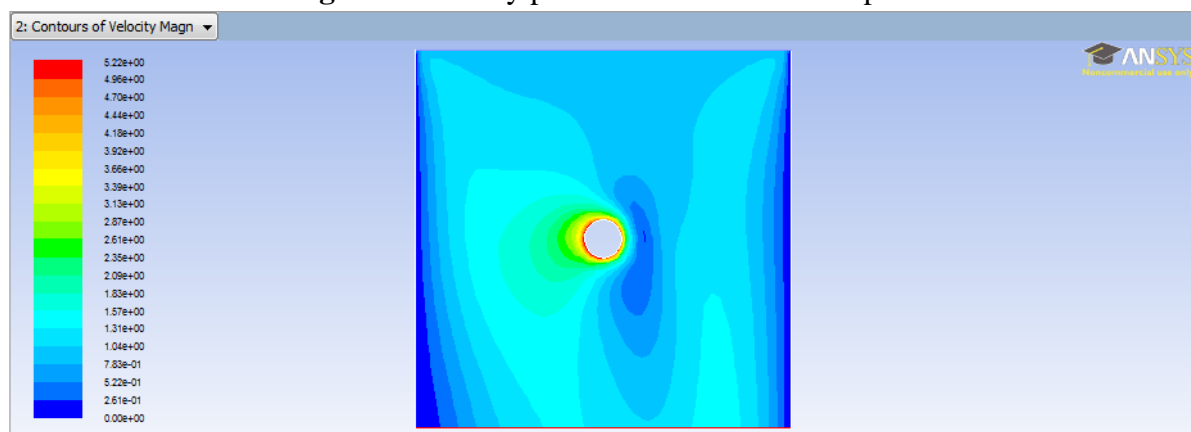
**Fig. 14** - Pressure profile at  $Re = 5$  and at 0rpm



**Fig. 15** - Pressure profile at  $Re = 5$  and at 100rpm



**Fig. 16** - Velocity profile at  $Re = 5$  and at 0rpm



**Fig. 17** - Velocity profile at  $Re = 5$  and at 100rpm

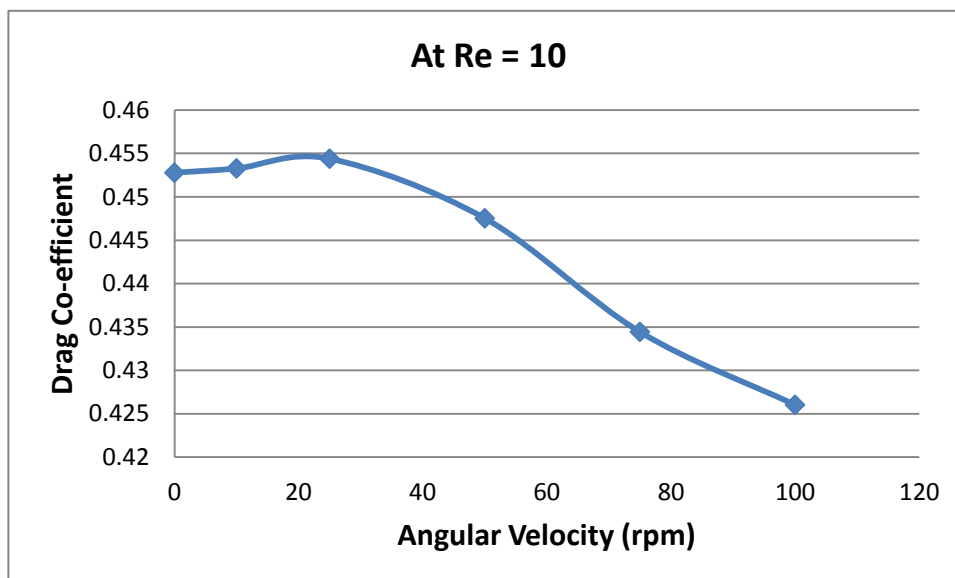


Now the value of Reynolds Number is changed to 10 and the values of drag co-efficient, with angular velocity varying from 0 rpm to 100 rpm is tabulated as follows :-

**Table 5.4 - Values of Drag Coefficient at Reynolds Number = 10 and different angular velocities :-**

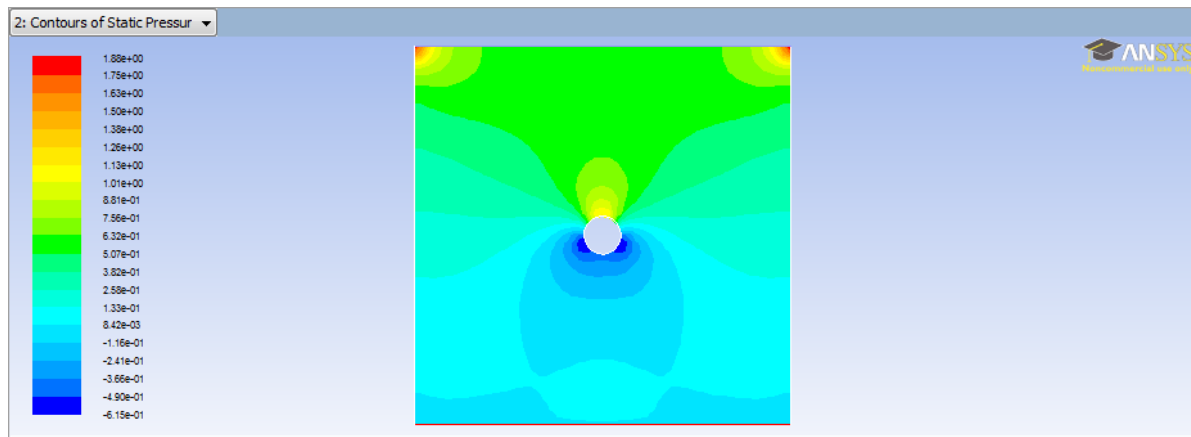
Angular velocity (rpm)	Drag coefficient (Cd)
0	0.45278
10	0.45327
25	0.45439
50	0.44752
75	0.43443
100	0.42602

The graph obtained between the angular velocity and drag co-efficient is plotted as follows :-

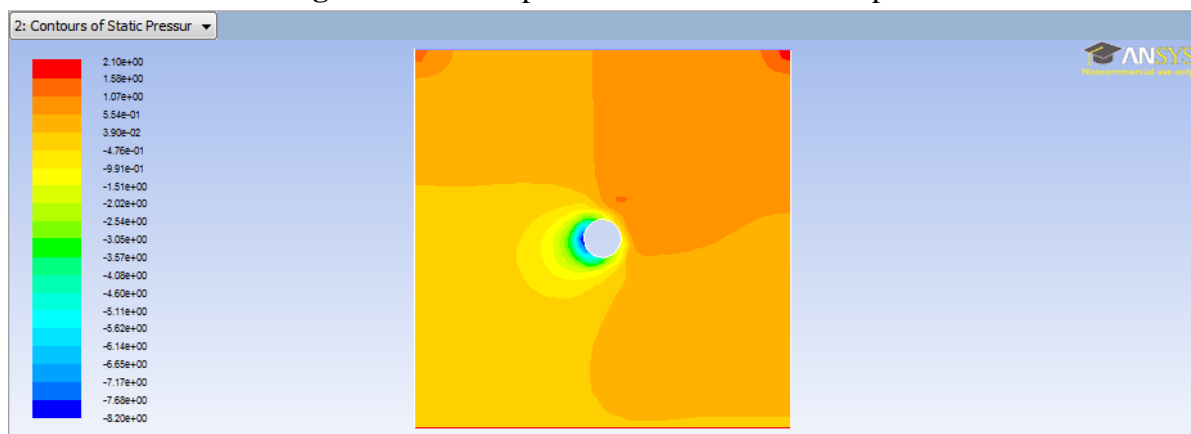


**Fig. 18 - Angular Velocity vs Cd graph at Re = 10**

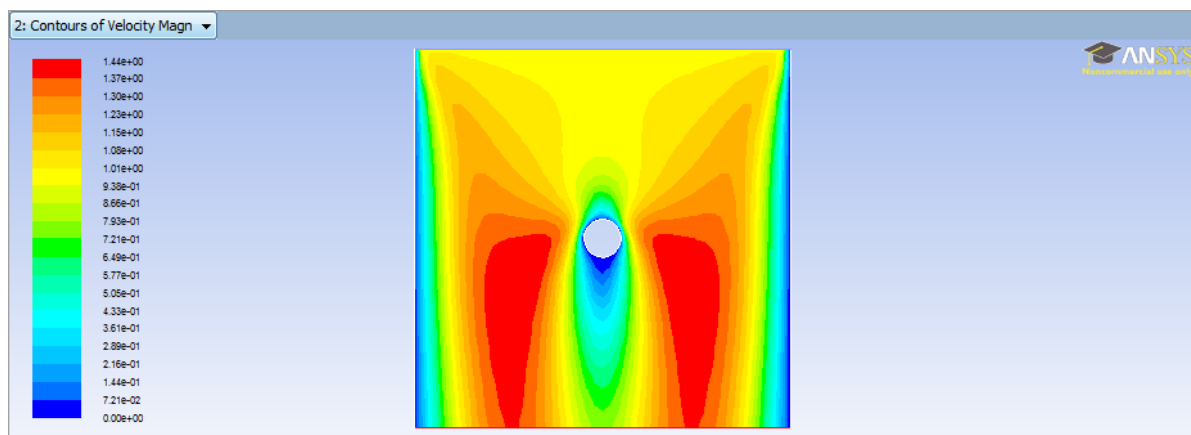
Now, the pressure profiles and the velocity profiles at 0 rpm and 100 rpm are observed to be as follows :-



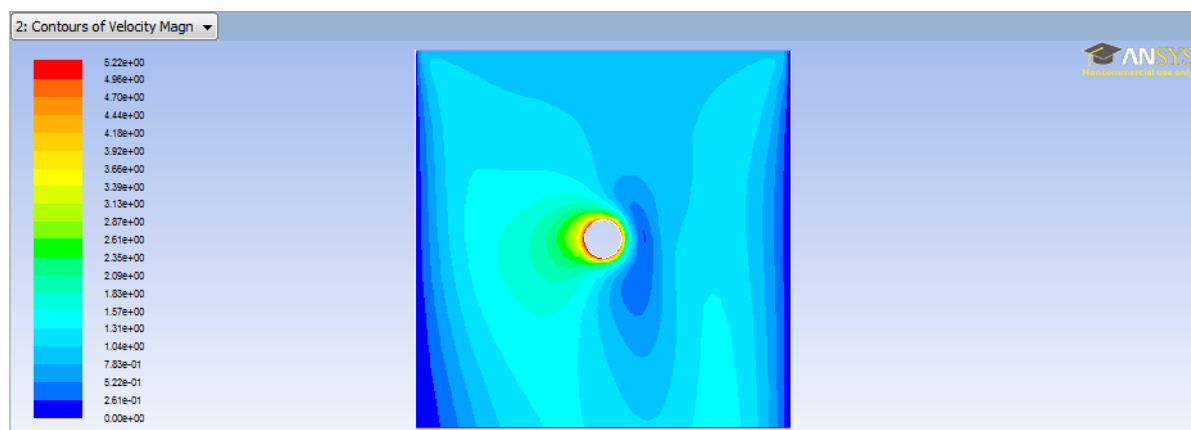
**Fig. 19** - Pressure profile at  $Re = 10$  and at 0rpm



**Fig. 20** - Pressure profile at  $Re = 10$  and at 100rpm



**Fig. 21** - Velocity profile at  $Re = 10$  and at 0rpm



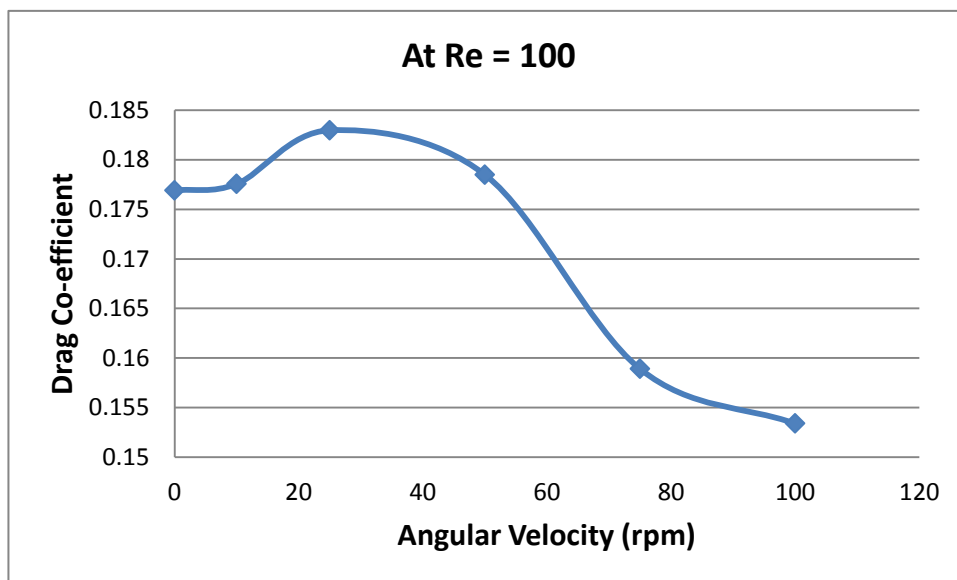
**Fig. 22** - Velocity profile at  $Re = 10$  and at 100rpm

Further, the value of Reynolds Number is changed to 100. The tabulation for this value is given as :-

**Table 5.5 - Values of Drag Coefficient at Reynolds Number = 100 and different angular velocities :-**

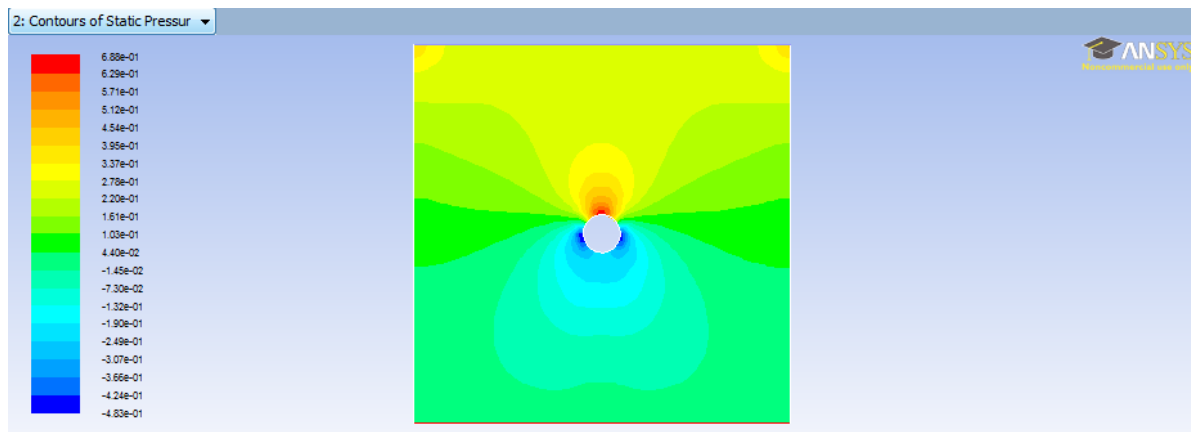
Angular velocity (rpm)	Drag coefficient (Cd)
0	0.17691
10	0.17757
25	0.18297
50	0.17849
75	0.15892
100	0.15341

The graph of this is observed to be as follows :-

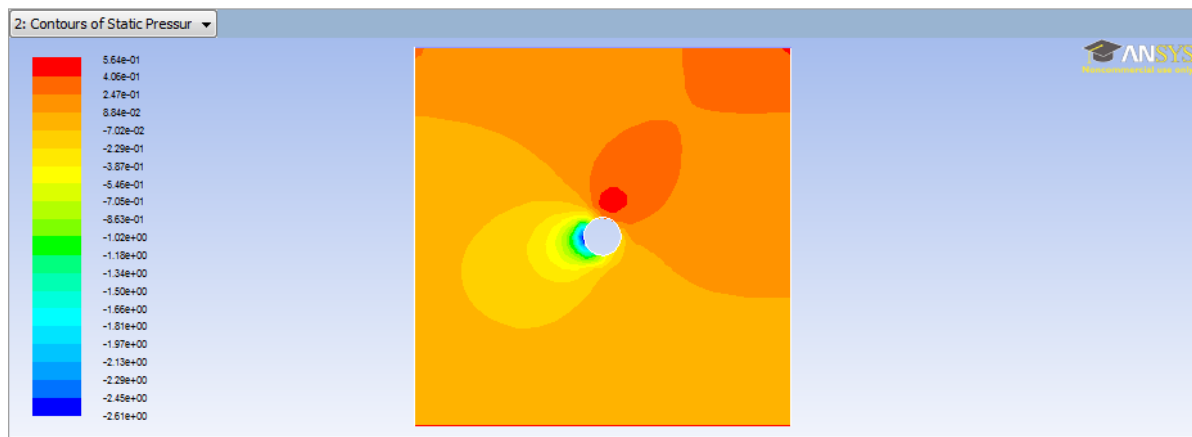


**Fig. 23 - Angular Velocity vs Cd graph at Re = 100**

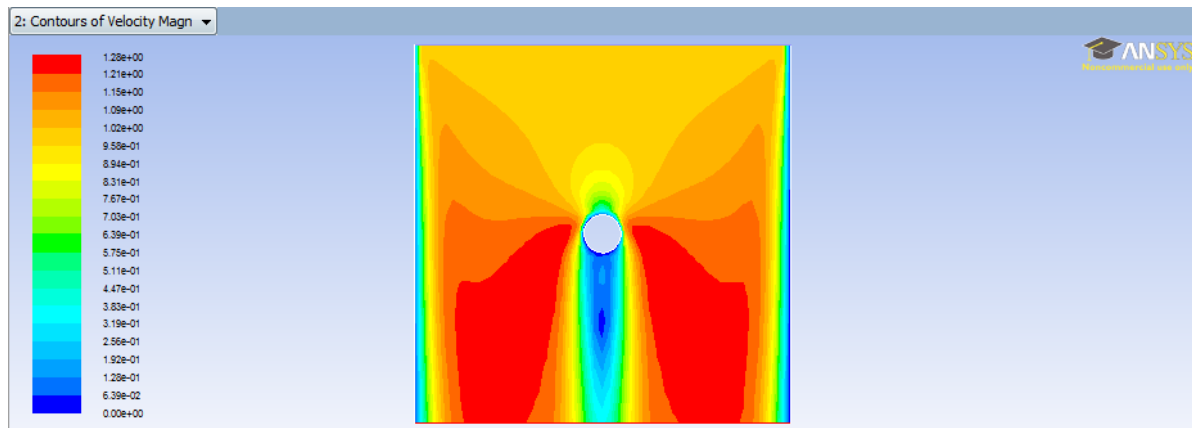
Further, the pressure and velocity profiles at 0 rpm and 100 rpm are obtained as follows :-



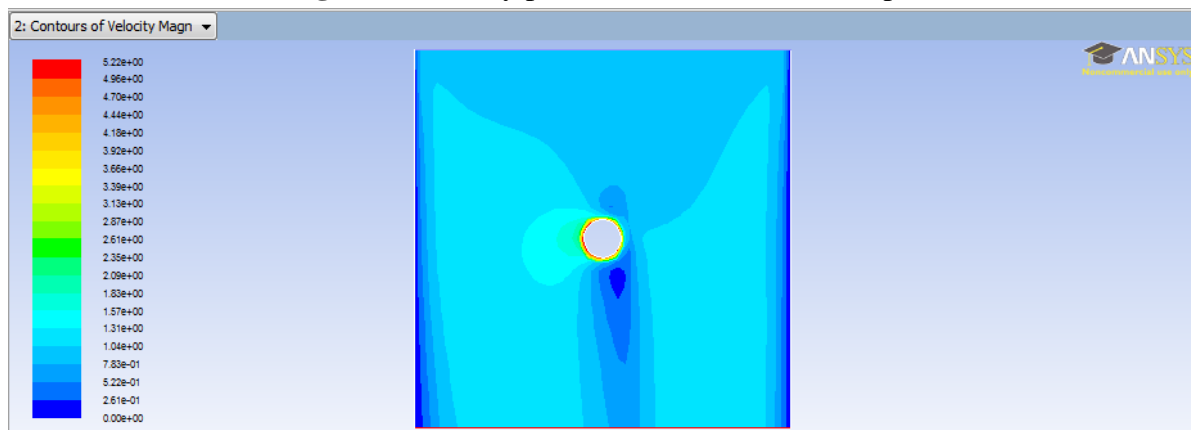
**Fig. 24** - Pressure profile at  $Re = 100$  and at 0rpm



**Fig. 25** - Pressure profile at  $Re = 100$  and at 100rpm



**Fig. 26** - Velocity profile at  $Re = 100$  and at 0rpm



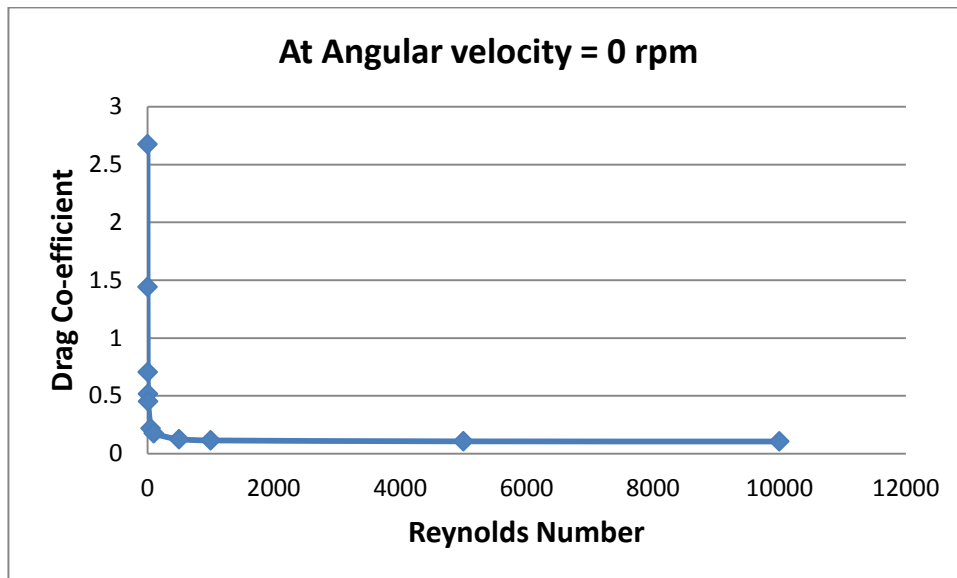
**Fig. 27** - Velocity profile at  $Re = 100$  and at 100rpm

Similarly, the drag coefficient is found out at various Reynolds Numbers. However, for the purpose of comparison between the values, the cylinder is assumed to be static. Hence, the value of angular velocity is kept at 0 rpm for the following tabulation :-

**Table 5.6 - Values of Drag Coefficient at various Reynolds Numbers :-**

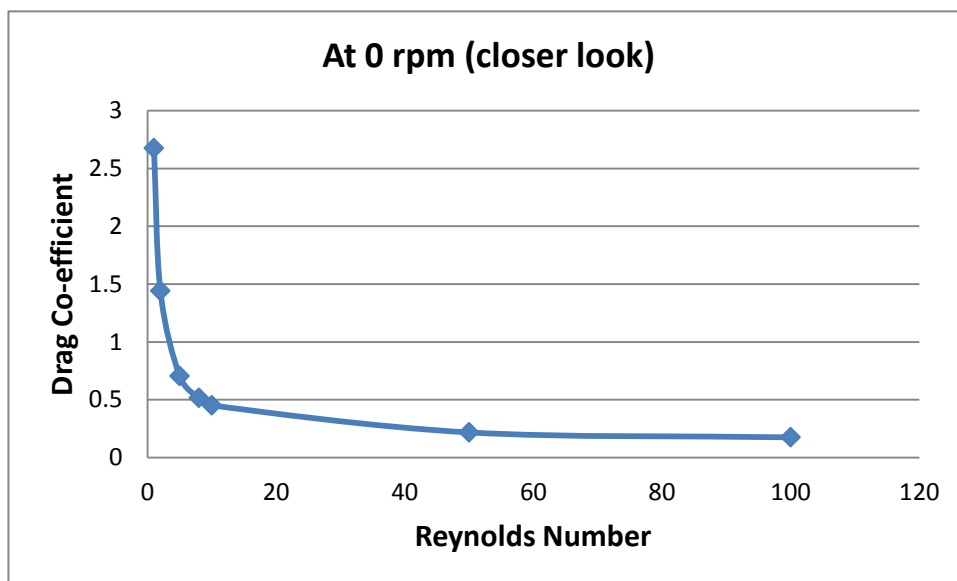
Reynolds Number	Drag coefficient
1	2.6756
2	1.4422
5	0.70599
8	0.51746
10	0.45278
50	0.21922
100	0.17691
500	0.12561
1000	0.11595
5000	0.10787
10000	0.10686

Hence, the graph between these values is observed to be as follows :-



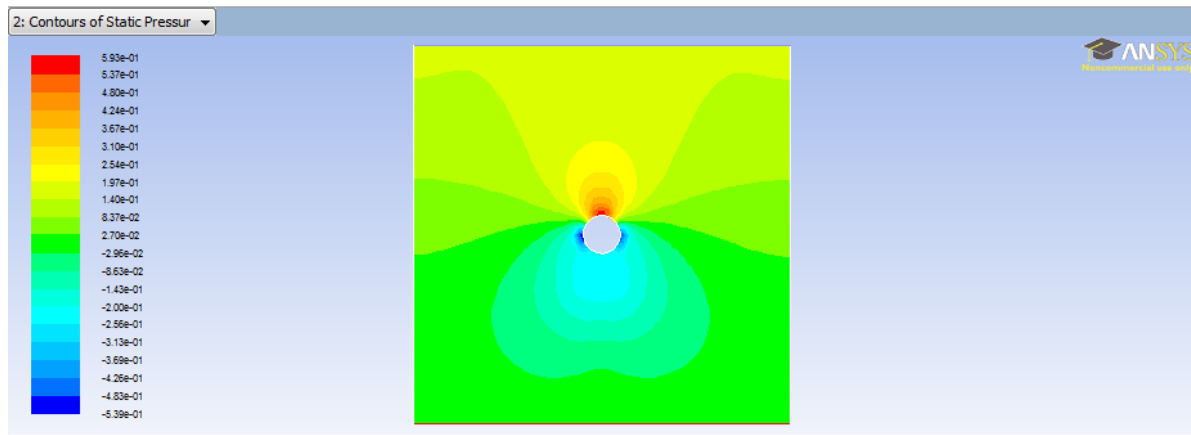
**Fig. 28 - Reynolds Number vs Cd graph at 0 rpm**

For a better observation for values of Reynolds Number less than 100, a separate graph is shown below :-

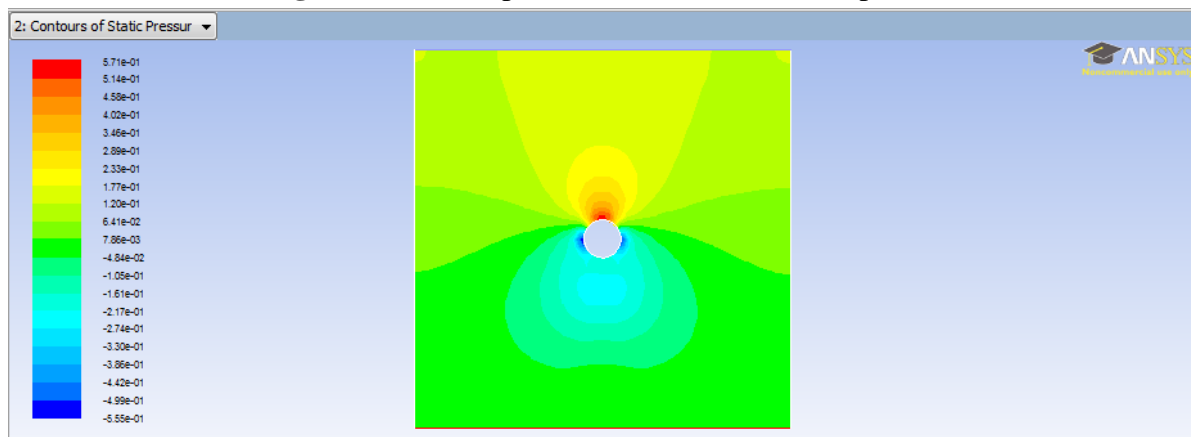


**Fig. 29 - Reynolds Number vs Cd graph at 0 rpm  
(closer look)**

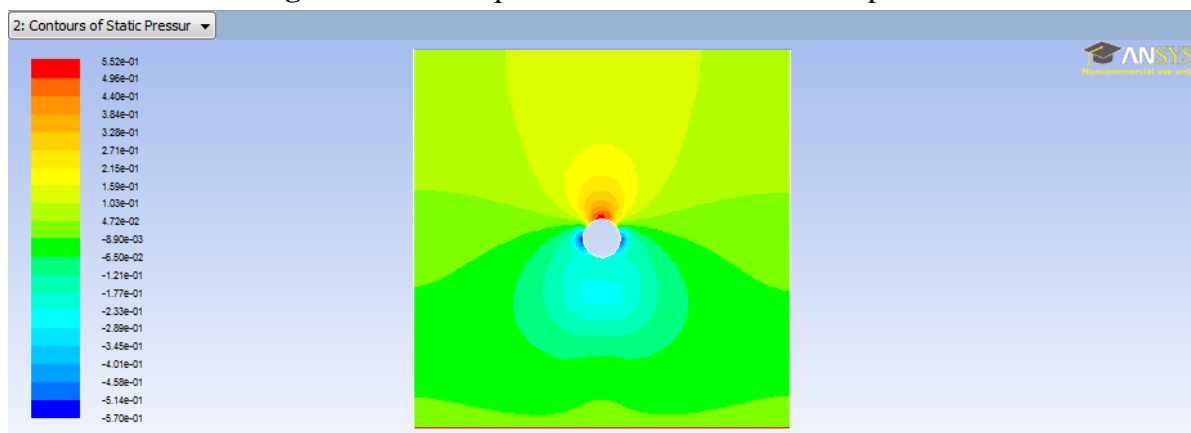
For better understanding of how the velocity and pressure profiles get affected with the change in Reynolds Number, the following profiles can be taken into account :-



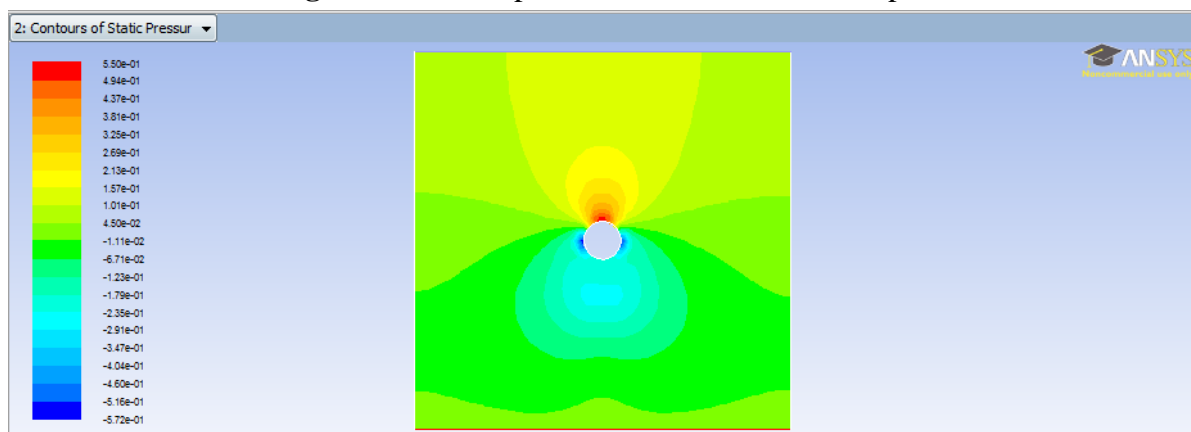
**Fig. 30** - Pressure profile at  $Re = 500$  and at 0rpm



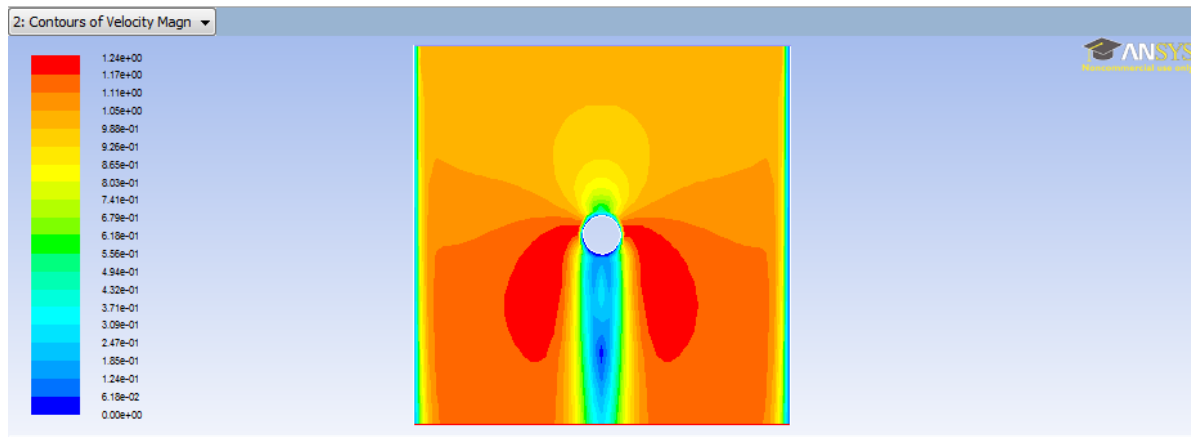
**Fig. 31** - Pressure profile at  $Re = 1000$  and at 0rpm



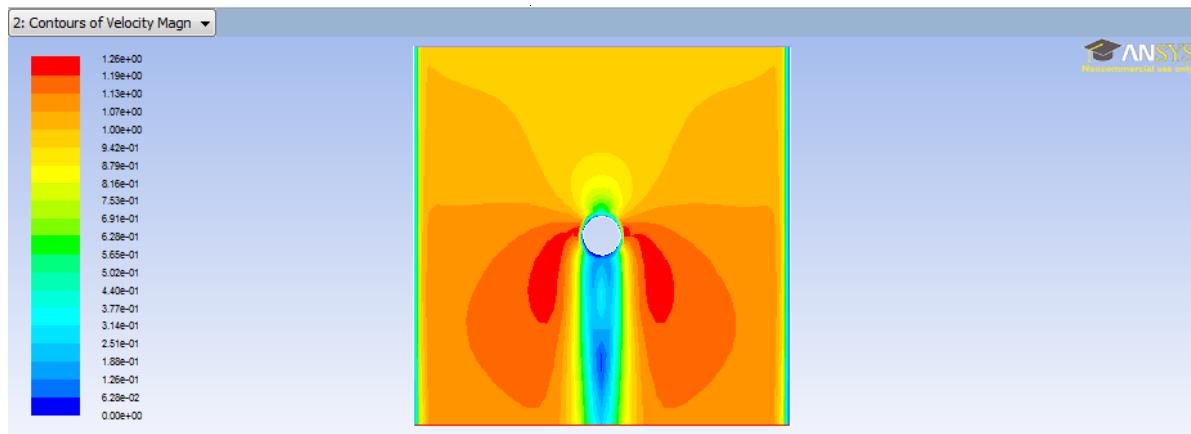
**Fig. 32** - Pressure profile at  $Re = 5000$  and at 0rpm



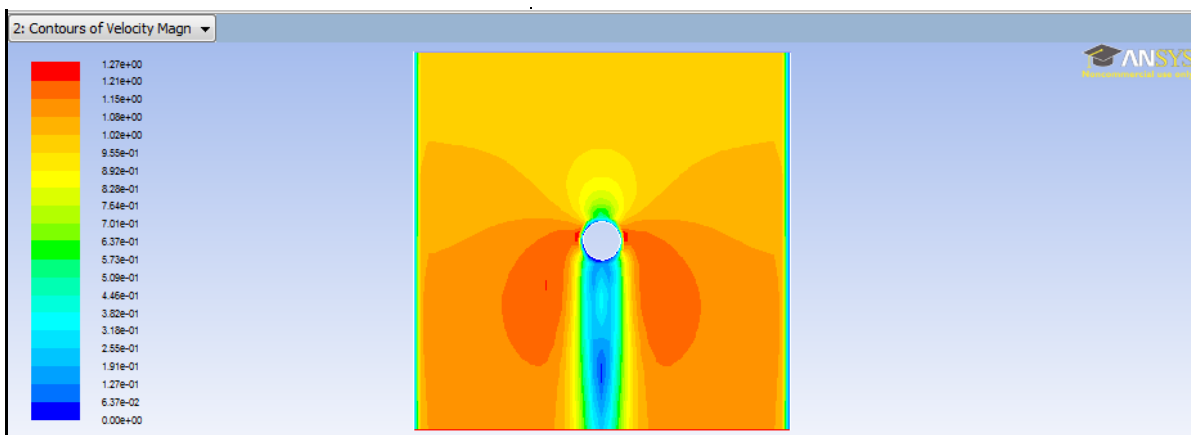
**Fig. 33** - Pressure profile at  $Re = 10000$  and at 0rpm



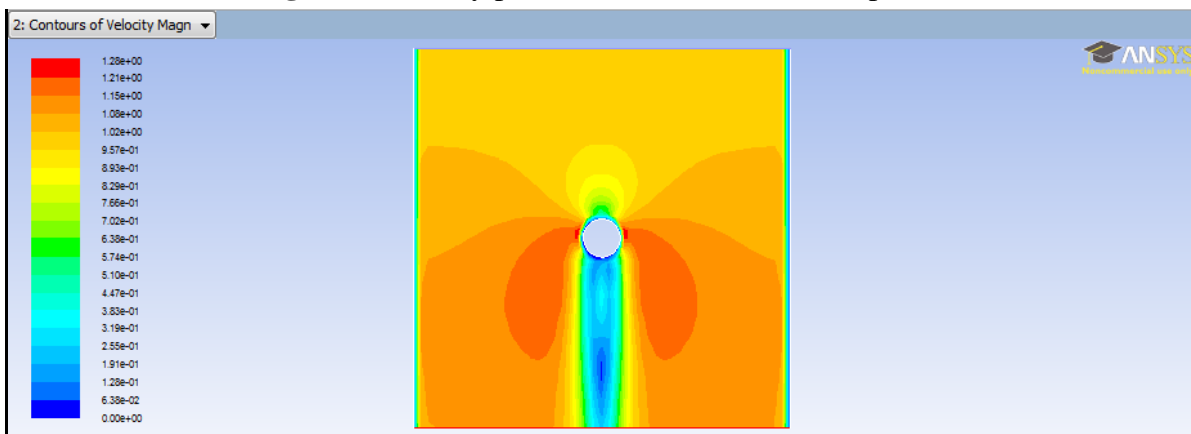
**Fig. 34** - Velocity profile at  $Re = 500$  and at  $0\text{rpm}$



**Fig. 35** - Velocity profile at  $Re = 1000$  and at  $0\text{rpm}$



**Fig. 36** - Velocity profile at  $Re = 5000$  and at  $0\text{rpm}$



**Fig. 37** - Velocity profile at  $Re = 10000$  and at  $0\text{rpm}$



# CHAPTER 6

## CONCLUSION

From the simulation performed, the following conclusions can be drawn :-

1. With the increase in Reynolds Number from 1 to 10, the drag coefficient shows a great decrease in its value.
2. However, at higher values of Reynolds Number, this decrease in drag coefficient decreases its pace.
3. At much higher value of Reynolds Number (above 1000), the value of drag coefficient shows negligible drop in its value.
4. The reason behind this behavior of drag coefficient is that Reynolds Number depends inversely on viscosity. More is the viscosity, more will be the area of contact between the fluid and the surface of the cylinder. Since the drag coefficient depends inversely on area of contact, hence, more area results in lesser value of drag coefficient.

Although the factor of velocity could also have been taken into concern, as Reynolds Number depends directly on velocity and hence, depends directly on area of contact. But here, we have assumed our velocity to be constant at 1m/s. Hence a better relation between the drag coefficient, area of contact, Reynolds Number and viscosity could be understood.

5. The effect of angular velocity is not much significant at lower values, but after certain value, it decrease the value of drag coefficient.
6. The velocity profiles are found to be quite symmetric when the angular velocity is at zero rpm i.e. the cylinder is static. Same is the case with the pressure profiles at zero rpm.
7. More the value of angular velocity, more is the difference in fluid flow velocity created on the opposite sides of the cylinder. One of the sides gets its path smoother and easier, with the motion of the cylinder favouring its flow. While on the other side, the motion gets a bit hindered and hence velocity decreases.

Thus, it can be concluded that with the increase in the angular velocity, the symmetry in the profile is lost.

8. From the velocity contours, it is clearly observable that at zero rpm, the fluid velocity around the cylinder tends to zero. Even in pressure profiles, the blue color around the cylinder is more prominent.
9. As the angular velocity increases, the velocity around the cylinder begins to increase. Hence, at 100rpm profiles, the red color becomes more prominent in case of

velocity. The pressure however retains its blue color.

10. Hence, it can be concluded that the angular velocity highly affects the fluid velocity, but does not bring drastic change in case of pressure profiles, throughout the cases considered throughout the simulation here.

## REFERENCES:

1. [http://en.wikipedia.org/wiki/Newtonian\\_fluid](http://en.wikipedia.org/wiki/Newtonian_fluid)
2. [http://en.wikipedia.org/wiki/Non-newtonian\\_fluid](http://en.wikipedia.org/wiki/Non-newtonian_fluid)
3. [http://en.wikipedia.org/wiki/Computational\\_fluid\\_dynamics](http://en.wikipedia.org/wiki/Computational_fluid_dynamics)
4. <http://en.wikipedia.org/wiki/Ansys>
5. McCormick, Barnes W. (1979): Aerodynamics, Aeronautics, and Flight Mechanics, John Wiley & Sons, Inc., New York, ISBN 0-471-03032-5, p. 24,
6. Milne-Thomson, L.M. (1973). Theoretical Aerodynamics. Dover Publications. ISBN 0-486-61980-X.
7. Hess J. L., Smith A. M. O. "Calculation of Potential Flow about Arbitrary Bodies". Progress in Aeronautics Sciences, Vol. 8, pp. 1-138, Bibcode (1967)
8. Daniel Guggenheim School of Aerospace Engineering. Retrieved 2007-07-28. <http://soliton.ae.gatech.edu/people/sruffin/nascart/>
9. Investors.com-<http://news.investors.com/article/593602/201112050805/top-stocks-include-intuitive-surgical-nuance.htm?Ntt=ansys-smartselect&p=3>
10. <http://money.cnn.com/magazines/fortune/fortunefastestgrowing/2010/index.html>
11. SEC Filing <http://anss.client.shareholder.com/secfiling.cfm?filingID=1193125-12-74501&CIK=1013462>
12. Bruschi Giancarlo, Nishioka Tomoko, Tsang Kevin, Wang Rick. A comparison of analytical methods - Drag coefficient of a cylinder [Water tunnel experiment] [Submitted on 21<sup>st</sup> March 2003]
13. Chaitanya, V., Parimal N., Ravindranath, G., Satish, K, B., Albert, K. Basics of computational fluid dynamics analysis (2003)
14. Ghoshdastidar S P. Computer simulation of flow and heat transfer. Tata McGraw-Hill Publishing Company Limited, New Delhi (1998)
15. Batchelor G. K. An Introduction to Fluid Dynamics. The Press Syndicate of the University of Cambridge , United Kingdom (1967)
16. [http://en.wikipedia.org/wiki/Navier\\_stokes\\_equation](http://en.wikipedia.org/wiki/Navier_stokes_equation)
17. Bakker, A. Applied computational fluid dynamics, Fluent Introductory notes. Fluent Inc. Lebanon. (2002)
18. [http://ethesis.nitrkl.ac.in/2336/1/CFD\\_Analysis\\_of\\_Phase\\_Holdup\\_Behaviour\\_in\\_a\\_Three\\_Phase\\_Fluidized\\_Bed.pdf](http://ethesis.nitrkl.ac.in/2336/1/CFD_Analysis_of_Phase_Holdup_Behaviour_in_a_Three_Phase_Fluidized_Bed.pdf)
19. [http://en.wikipedia.org/wiki/Drag\\_coefficient](http://en.wikipedia.org/wiki/Drag_coefficient)

21. <http://www.grc.nasa.gov/WWW/k-12/airplane/shaped.html>
22. <http://www.grc.nasa.gov/WWW/k-12/airplane/shaped.html>
23. <http://www.grc.nasa.gov/WWW/k-12/airplane/presar.html>
24. Stokes, George. "On the Effect of the Internal Friction of Fluids on the Motion of Pendulums". Transactions of the Cambridge Philosophical Society Vol. 9 (1851), pp. 8-106
25. Reynolds, Osborne. "An experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous and of the law of resistance in parallel channels". Philosophical Transactions of the Royal Society, Vol.174 (0) (1883), pp. 935–982.
26. Rott, N. "Note on the history of the Reynolds number". Annual Review of Fluid Mechanics, Vol. 22 (1) (1990), pp. 1–11
27. Reynolds Number, <http://engineeringtoolbox.com>